



National Aeronautics and  
Space Administration

# **BLADE LOSS TRANSIENT DYNAMICS ANALYSIS**

## **Final Report Volume II TASK II - TETRA 2 USER'S MANUAL**

**November 1986**

by

Gerald Black  
Vincente C. Gallardo

General Electric Company  
Aircraft Engine Business Group  
Cincinnati, Ohio 45215

Prepared For

**National Aeronautics and Space Administration**

(NASA-CR-179633) BLADE LOSS TRANSIENT  
DYNAMICS ANALYSIS, VOLUME 2. TASK 2: TETRA 2  
USER'S MANUAL Final Report (General  
Electric Co.) 204 p Avail: NTIS HC  
A10/MF A01

N88-10792

Unclas

CSCL 21E G3/07 0106712

**NASA Lewis Research Center  
Contract NAS3-24381**

1. Report No. NASA CR-179632		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle NASA BLADE LOSS PROGRAM TURBINE ENGINE STEADY STATE RESPONSE ANALYSIS VOLUME 2: TETRA 2 USER'S MANUAL				5. Report Date NOVEMBER 1986	
				6. Performing Organization Code	
7. Author(s) Gerald R. Black Vincente C. Gallardo				8. Performing Organization Report No.	
				10. Work Unit No.	
9. Performing Organization Name and Address General Electric Company Aircraft Engine Group 1 Newuman Way Cincinnati, Ohio 45215				11. Contract or Grant No. NAS3-24381	
				13. Type of Report and Period Covered Contract Final	
12. Sponsoring Agency Name and Address NASA - Lewis Research Center Materials, Structure, and Grants Section 21000 Brook Park Road Cleveland, Ohio 44135				14. Sponsoring Agency Code	
15. Supplementary Notes					
16. Abstract  <p>This is the user's manual for the TETRA 2 Computer Code, a program developed in the NASA-Lewis Blade Loss Program. TETRA 2 calculates a turbine engine's dynamic structural response from applied stimuli. The calculation options are: (1) Transient Response, and (2) Steady State Forced Response. Based on the method of modal syntheses, the program allows the use of linear, as well as nonlinear connecting elements. Both Transient and Steady State options can include:</p> <p style="padding-left: 40px;">Flexible Bladed Disk Module, and Nonlinear Connecting Elements (including dead band, hardening/softening spring).</p> <p>The Transient option has the additional capability to calculate response with a squeeze film bearing module.</p> <p>TETRA 2 output is summarized in a plotfile which permits post processing such as FFT or graphical animation with the proper software and computer equipment.</p>					
17. Key Words (Suggested by Author(s)) TETRA, Rotor Dynamics, Turbine Engine, Transient Response, Steady State, Harmonic Balance, Unbalance, Forced Vibration, Blade Out, TETRA 2.				18. Distribution Statement	
19. Security Classif. (of this report) Unc.		20. Security Classif. (of this page) Unc.		21. No. of Pages	
				22. Price*	

\* For sale by the National Technical Information Service, Springfield, Virginia 22161

## TABLE OF CONTENTS

<u>Section</u>	<u>Page</u>
1.0 INTRODUCTION AND SUMMARY	1
2.0 TETRA 2 INPUT	5
2.1 Input Restart File	5
2.2 Primary Input File	6
2.3 TETRA 2 Input Sheets	7
2.3.1 Overall Input (Type A)	8
2.3.2 Physical Points Not Located On Modal Subsystems (Type B)	12
2.3.3 Modal Subsystem Input (Type C-1 through C-16)	14
2.3.3.1 Vertical and Horizontal Plane Subsystems (Type C-1 through C-3)	16
2.3.3.2 Rigid Body Subsystems (Type C-4 through C-6)	20
2.3.3.3 Case Torsional Subsystem (Type C-7 through C-9)	24
2.3.3.4 Pylon Subsystem (Type C-10 through C-12)	28
2.3.3.5 Flexible Bladed Disk Subsystems (Type C-13 through C-16)	32
2.3.4 Physical Connecting Element Input (Type D-1 through H-3)	37
2.3.4.1 General Spring-Damper Element (Type D-1 and D-2)	38
2.3.4.2 Space Link-Damper Element (Type E)	41
2.3.4.3 Rub element (Type F)	43
2.3.4.4 Engine Support-Links Element (Type G-1 through G-3)	45
2.3.4.5 Uncoupled Point Spring-Damper Element (Type H-1 and H-2)	49
2.3.4.6 Squeeze Film Damper Element (Type H-3)	52
2.3.5 Restart, Time Integration, and Print/Plot Multiples (Type I-1)	54
2.3.6 Steady State Nonlinear Analysis Input (Type I-2)	57
2.3.7 Rotor Speed and Rate Input (Type J-1 and J-2)	59
2.3.8 Applied Loads (Type K-1 through M-2)	62
2.3.8.1 Unbalance Loads (Type K-1 and K-2)	62
2.3.8.2 $P \cos(\omega t)$ and $P \sin(\omega t)$ Loads (Type L-1)	65
2.3.8.3 $P \cos(\omega t + \phi)$ Loads (Type L-2)	67
2.3.8.4 Time-Force History Loads (Type M-1 and M-2)	69
2.3.9 Gyroscopic Loads (Type N)	72
2.3.10 Data For Plot File (Type P-1 and P-2)	74

## TABLE OF CONTENTS (Continued)

<u>Section</u>	<u>Page</u>
3.0 TETRA 2 OUTPUT	77
3.1 Output Restart File	77
3.2 Output Plot File	78
3.3 Printed Output	88
4.0 PROGRAM STRUCTURE AND SUBROUTINES	89
5.0 STEADY STATE ANALYSIS VERIFICATION	103
5.1 Degenerate Cases: Closed Form Solutions	104
5.1.1 Single Degree-of-Freedom	104
5.1.2 Two Degree-of-Freedom	104
5.2 Nonlinear Systems: Simple Models	110
5.2.1 Single Degree-of-Freedom: Nonlinear Spring	110
5.2.2 Planar Rigid Beam with Two Nonlinear Bearings	113
5.3 Demonstrator Model	118
5.3.1 Input and output	118
5.3.2 Frequency Sweep with No Rub Element Test Cases	
5.3.3 Single Frequency with One Rub Element Test Cases	
5.3.3.1 No Rub	
5.3.3.2 Continual Rub with Dead Band Equal to Zero	
5.3.3.3 Continual Rub with Dead Band Not Equal to Zero	
5.3.3.4 Intermittent Rub	
6.0 REFERENCES	

## 1.0 INTRODUCTION AND SUMMARY

This user's manual for the TETRA 2 program provides the requisite inputs and description to perform a steady state response analysis of a turbine engine. In addition, since TETRA 2 was developed with the dual capability of steady state response and the transient analysis of the original program, inputs for both options have been consolidated. This includes the common modeling procedures (and basic inputs) and the separate inputs unique to each option. Hence TETRA 2 is a single program wherein the analytical and computational distinctions of the two options are transparent to the user.

The transient option has all the capabilities of the original, including the later modules:

- (1) Flexible Bladed Disk
- (2) Squeeze Film Bearing

In addition, the scope of the transient analysis has been increased so that it now contains the following enhancements initially developed for the steady state analysis: improved rub element (with the cubic nonlinear factor), structural damping capability (applies for physical connecting element types 1, 2, 4, and 5), and new printout options.

Also, changes in the coding of TETRA 2 were made to improve the efficiency of transient analyses. For most transient analysis runs, TETRA 2 is more than three times as fast as the earlier TETRA program.

With the exception of the squeeze film bearing module, the added elements are also available in the steady state analysis. This exception was made because the squeeze film bearing is valid only for circular motion, whereas the steady state analysis is generally for elliptical orbits - which can exhibit large eccentricity or ellipticity.

The driving motivation for the development of the steady state analysis was the need to calculate the steady unbalance response as a function of speed more directly. At the same time, it was needed to overcome the axi-symmetric limitations of current critical speed computer programs, and provide a computational method to calculate the realistic motions of actual engines. The non-isotropy in turbine support structures, such as those inherent in engine pylons and typical engine support hard points, require that the permissible engine motions are also non-isotropic. In other words, engine orbital motions are really elliptical, not circular.

Current methods have required two analyses of critical speeds or steady state response - which are both axi-symmetric. In one, the structural stiffnesses correspond to the vertical plane stiffness; in the other, the horizontal plane stiffness. Additionally, this procedure does not truly represent the cross-axis (cross-planar) coupling intrinsic to gyroscopic moments on rotating and vibrating disks and rotors.

TETRA 2 therefore was developed to overcome the shortcomings of standard steady

state analysis programs, at the same time providing new capabilities. These of course include the general nonlinear rub element (incorporating a deadband and a hardening or softening spring), the flexible bladed disks, and the modular design of the computer program. The latter allows the user to develop other connecting element modules or extend the basic structural components to accommodate more than four structural elements.

The consolidation of the transient and the steady state options provide one computational method to perform most dynamic analyses of realistic turbine engine structural systems. Even though emphasis is on turbine engines, TETRA 2 can also be used to perform dynamic analysis of non-turbine structures such as aircraft, buildings, or other structures which can be modeled with component structural components (and normal modes) connected by connecting elements.

The charts presented in figures 1-1 and 1-2 graphically summarize the capabilities and operations of TETRA 2. The latter flow chart delineates TETRA 2 in its two modes or options. The basic structural modeling and input information are common to both, including those for the connecting elements and other modules. Branching to either option is governed by the natural differences in the analyses: the transient option input/output include those that depend on time, such as rotor speed history, blade loss onset, number of time intervals, etc.; the steady state analysis input/output are in the frequency domain so that this branch requires information dependent on excitation frequency such as: frequency sweep, initial iteration solution trials, and others.

Finally, due to the voluminous output of the transient solution, and in the case of critical speed calculations, of the steady state, a plot file output is recommended. This avoids the extensive printed output, at the same time providing quick and easily interpretable presentation of the results. Such a plot file is especially indispensable for other types of output processing, such as animation or FFT analysis of transient data, and the frequency response plot presentation of the steady state output.

This volume contains the updated inputs for TETRA 2, combining the transient and steady state options. In addition explanations of the input variables and output are also included.

Figure 1-1. Structural Breakdown for TETRA Analysis

# Structural Breakdown for TETRA Analysis

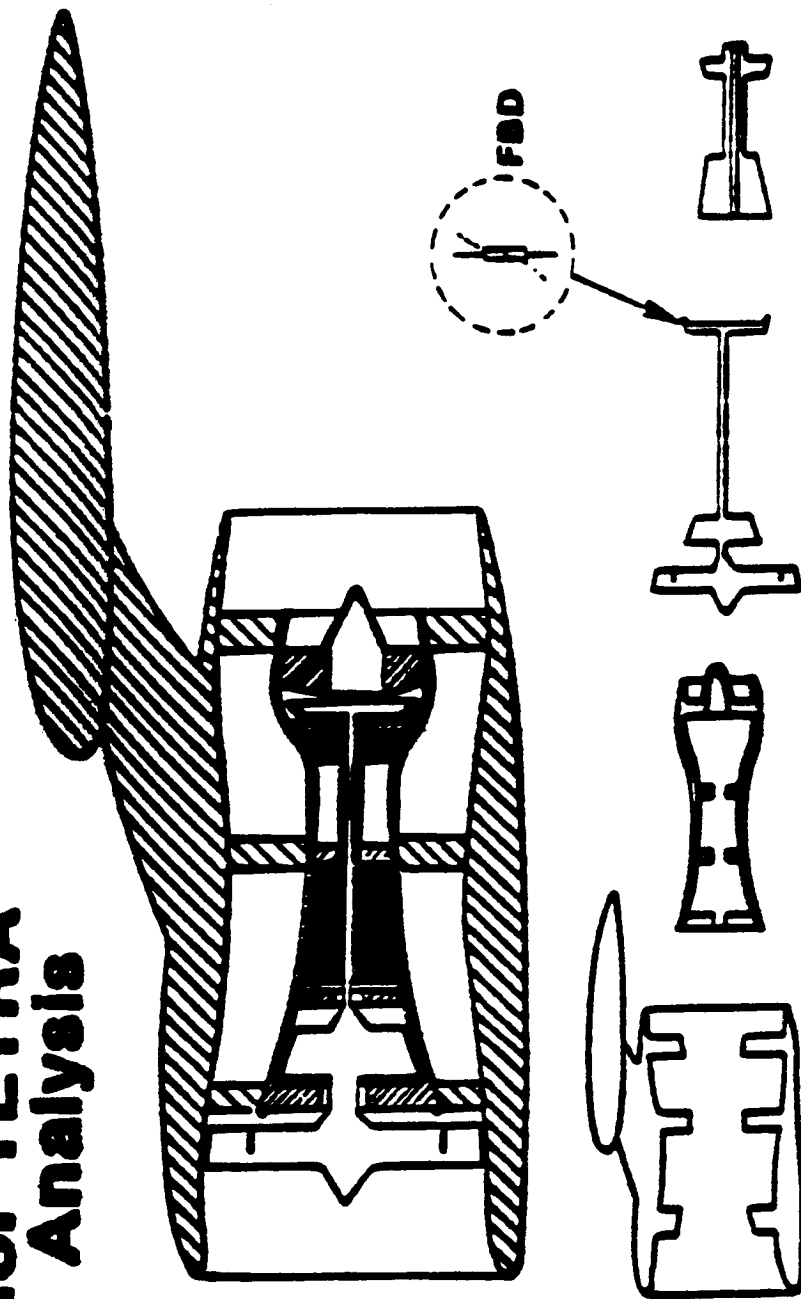
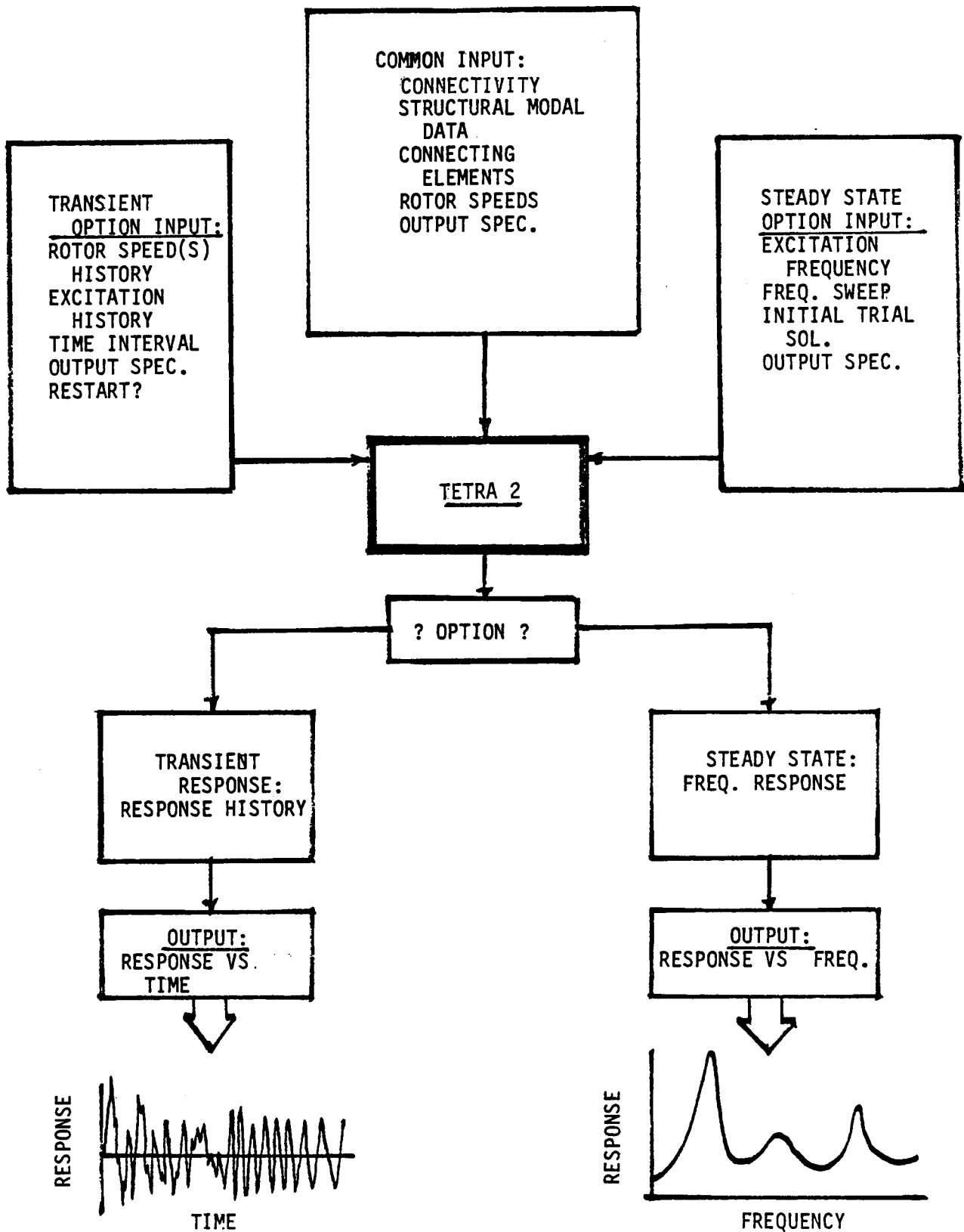


FIGURE 1-2 TETRA 2 SCHEMATIC DIAGRAM





## 2.0 TETRA 2 INPUT

### 2.1 Input Restart File

Transient analysis TETRA runs can be restarted using a restart file from an earlier TETRA run. A transient analysis run starts at time = 0 and marches forward in time. To begin a transient analysis run at a restart time (greater than 0), generalized coordinates and other data is needed for the restart time, and the generalized coordinates are also needed for the two time steps prior to the restart time. The restart file is the means of communicating this data calculated by the earlier TETRA run to the restarted TETRA run. For more information on restarting TETRA transient analysis runs, see the discussion of the type I-1 input sheet.

Steady state analysis runs cannot be restarted using a restart file from an earlier TETRA run. Restarting is unnecessary for a steady state analysis run, because steady state analyses can be started at any forcing frequency, and no knowledge of previous forcing frequencies that may or may not have been run is needed. The only possible exception to this is that the user might want to input an initial guess for the rub element relative displacements (the unknowns in the iterating equations) which was obtained from the printout at a given forcing frequency of a previous TETRA run. However, if this is the case the user can easily enter this initial guess in the primary input file (input sheet I-3). Furthermore, an initial guess is not required. For more information regarding an initial guess, see the discussion of the type I-3 input sheet.

The input restart file is read in using file code 22. Therefore, if the run is a restart run, the user must include a line or two in his or her JCL (job control language) identifying the restart file and associating it with file code 22.

The restart file is an unformatted sequential file. It is a short file and takes up very little disk space.

## 2.2 Primary input file

The primary TETRA input file consists of namelist input. The file code for the primary input file is the standard 05 file. See section 2.3 for a discussion of the namelist input sheets for this file.

As in the original TETRA program, the namelist input is divided into sections of which there are four types named LIST1, LIST2, LIST3, and LIST4 (must be arranged in that order in the input file). Each section must begin with a \$ followed immediately by the section name (that is, \$LIST1, \$LIST2, \$LIST3, or \$LIST4). Following this comes the input variable names and their values (which may be entered in any order). On the Honeywell computer, each section must end with a \$. On the Cray computer, either a \$ or a \$END can be used to signal the end of a section. The namelist input should begin in column 2 of each line.

The LIST1 namelist input is for the overall input and input for points not located on the modal subsystems. There must be one and only one LIST1 section. Next come the LIST2 namelist sections. One LIST2 section is required for each modal subsystem (there must be at least one modal subsystem and at most 13). The LIST3 namelist sections (if any) come next. There is one LIST3 section for each physical connecting element. There may be zero physical connecting elements. The current upper limits are a maximum of 5 type 1 physical connecting elements, 10 of type 2, 10 of type 3, 4 of type 4, 25 of type 5, and 5 of type 6. (Incidentally, the upper limits for the program variables may change in the future as the program evolves. A listing of the current upper limits is included in the printed output from a TETRA run just following the listing of the input file, except when the short form of the printed output is requested by entering IOUT = 0 on the type A input sheet). Finally, there must be one and only one LIST4 section. The LIST4 section includes restart input, time integration input, print and plot multiples, steady state nonlinear analysis input, rotor speed and rate input, load input, and plot file input.

For a sample input file listing, see section 5.3.1. A listing of the input file for the demonstrator model is imbedded near the beginning of the printed output for the demonstrator model in this section.

### 2.3 TETRA 2 Input Sheets

The TETRA 2 program has 44 namelist input sheets. However, usually many of the input sheets are not needed, and the user just includes those input sheets that apply for his or her model. Unless otherwise stated on the input sheet, the input data is common to both transient and steady state analyses. If the input is specific to a transient or to a steady state analysis, this is so noted on the input sheet.

Examining the TETRA 2 input sheets, one can see that about 80% of the TETRA input (for instance, the usually lengthy modal subsystem input) is common to both transient and steady state analysis runs. Another 10% of the TETRA 2 input pertains specifically to a transient analysis run. Most of the input in these two categories was present in the earlier TETRA program (see references 1 and 2). Finally, about 10% of the TETRA input is specific to a steady state analysis run. This input is all new with TETRA 2, of course, since the earlier TETRA program could only perform transient and not steady state analyses.

Three enhancements were added to the TETRA 2 input that were made to apply for either transient or steady state analyses. These are: the rub element cubic nonlinear factor AK (on type F input sheet), structural damping frequency option ISF (on type A input sheet), and printout option IOUT (on type A input sheet).

The TETRA 2 input was set up such that so that the same input file could easily be used for either a transient or steady state analysis. Simply include in this input file both the input that is specific to a transient analysis and the input that is specific to a steady state analysis. When a transient analysis is run, TETRA 2 just ignores the input that is specific to a steady state analysis, and likewise, when a steady state analysis is run, TETRA 2 ignores the input that is specific to a transient analysis. Then you need only change the value of input variable NTYPE (type A input sheet) to tell TETRA 2 whether the run is for a transient or steady state analysis. A transient analysis is the default if input variable NTYPE is not included.

It is emphasized that the input for TETRA 2 is entirely consistent with that of the earlier TETRA program. In fact, if a user has an input file that he or she ran on the original TETRA program, that input file, without any modifications, will still work fine for TETRA 2, producing the same transient analysis run results as were produced by the original TETRA program. If the user then wants to use this input file for a steady state analysis run, he or she will have to add the variable NTYPE = 1 (type A input sheet) to tell the program that the run is for a steady state analysis and a small amount of additional input that is needed for a steady state analysis.

Many of the input sheets to the earlier TETRA program remain the same for TETRA 2. However, several input sheets to the earlier TETRA program were modified, and several others are new to TETRA 2. Although lots of this material is redundant of the earlier TETRA manuals (references 1 and 2), it was decided to discuss all the TETRA 2 input sheets in this manual, so that the user would only have to refer to one manual regarding the input sheets.

### 2.3.1 Overall Input (Type A Input Sheet)

Except for the possible presence of several lines of JCL (job control language), the first line on the input file must be \$LIST as shown on the type A input sheet. This line signals the beginning of the LIST1 namelist input section, which is comprised of type A and type B input sheets. Each input file must have one LIST1 namelist section.

Next, the variables NAME, ADDRESS, IDENT1, and IDENT2 are used to enter the user's name, address (building, mail drop, bin number, etc.), and two lines of description for the run. Up to 60 characters may be included between the apostrophes for each of these variables. These variables are not required, so one or more of them may be omitted if desired. If present, the information entered via these variables is reproduced on the first page (title page) of the printed output.

Following this, the analysis type flag is entered. If NTYPE = 0, or if this variable is omitted, a transient analysis is performed. If NTYPE = 1, a steady state analysis is performed.

Next comes the structural damping frequency option ISF. Structural damping is an enhancement which was added to TETRA 2 and was not present in earlier versions of TETRA. This feature can be used either for transient or steady state analyses. Structural damping may be used for any of the linear (type 1, type 2, type 4, or type 5) physical connecting elements, but is not available for the nonlinear physical connecting elements (type 3 or type 6).

The presence or absence of the ISF input variable is not what determines whether structural damping is present, so if the user does not want structural damping, it does not matter whether this variable is entered or not or what value it is given. Rather, the user signals that he or she wants structural damping for a given physical connecting element by entering a non-zero value for QELEM (Q-factor) and by omitting QFREQ (the selected frequency) on the input sheets for that element (see the input sheets for the physical connecting elements, types D-1 through H-3).

If the user has so indicated that structural damping is desired for a particular element, the program then uses the entered Q-factor and the frequency defined by the ISF value to calculate the damping. If ISF equals 1 or is omitted, then the independent rotor speed is used regardless of whether the run is a transient or a steady state analysis. (If you don't know what is meant by the independent rotor, read the discussion of the type J-1 and J-2 input sheets). However, if the user enters ISF = 2, the independent rotor speed is still used if the run is a transient analysis run, but the steady state forcing frequency is used if the run is a steady state analysis run. (For steady state runs, the steady state forcing frequency might be the same as the independent rotor speed, but is different from the independent rotor speed when defined by the BW, FW, and SW variables on input sheet L-2).

TETRA  
OVERALL INPUT

Enter the following name, address, and run identification input (maximum of 60 characters between apostrophes):

2

\$LIST1  
NAME='\_\_\_\_\_,  
ADDRESS='\_\_\_\_\_,  
IDENT1='\_\_\_\_\_,  
IDENT2='\_\_\_\_\_,'

Enter the analysis type flag NTYPE:

NTYPE=\_\_\_\_, 0 = Transient analysis (assumed value)  
          1 = Steady state analysis

Enter the structural damping frequency option ISF, which defines the frequency to be used along with the Q-factor to calculate the structural damping (if present) for type 1, type 2, type 4, or type 5 physical connecting elements. ISF applies for each physical connecting element for which the user inputs a non-zero value for QELEM (Q-factor) and omits QFREQ (the selected frequency) (see the input sheets for the physical connecting elements).

ISF=\_\_\_\_, 1 = Independent rotor speed used for both steady state and transient analyses (assumed value)  
          2 = Steady state forcing frequency used for steady state analyses and independent rotor speed used for transient analyses

Enter the printout option IOUT, which determines what output is printed.

IOUT=\_\_\_\_, 0 = Short form of the printout  
          1 = Standard form of the printout  
          2 = Long form of the printout

If the user inputs both QELEM (the Q-factor) and QFREQ (the selected frequency) on the input sheets for the given physical connecting element, then the program uses the entered Q-factor and entered frequency to calculate the damping for the element (overriding the frequency defined by the ISF variable). If this is the case, the damping is non-structural, since the constant frequency defined by QFREQ is used regardless of variations in the independent rotor speed or the steady state forcing frequency. The ability to enter non-structural damping in this manner was present in the earlier versions of TETRA.

The final variable entered on the type A input sheet is the printout option IOU<sub>T</sub>. Three possible values (0, 1, or 2) may be entered for this variable, but the value of 1 (standard form of the printout) usually is best (1 is also the default if this variable is omitted). A value of 1 provides a formatted listing of the input (very useful in debugging one's input) and provides all the information that the user is normally interested in for the requested time steps (for a transient analysis) or the requested forcing frequencies (for a steady state analysis). This includes the physical displacements, physical forces, and all other essential data.

If the user enters IOU<sub>T</sub> = 0, the short form of the printout is used. The short form of the printed output contains what the standard form of the output does minus the verbatim listing of the input file, the listing of the current program limits for various variables, and the formatted listing of the input as interpreted by the program. With the short form of the output, the title page is printed, followed by the same output for each time step (if a transient analysis) or for each forcing frequency (if a steady state analysis) that is printed for the standard output form.

If the TETRA program detects any input errors, it prints an error message exactly at the point in the formatted listing of the program input that caused the error. This is highly useful to the user in figuring out what was wrong. If the user inputs IOU<sub>T</sub> = 0, the error message is still printed out, but since the formatted listing of the program input has been suppressed, the user sometimes is left scratching his head trying to figure out exactly where in the input the error occurred. Thus, it is recommended that the user not enter IOU<sub>T</sub> = 0 until after the program has been successfully run and the input thoroughly verified. Once the user is sure that the input is okay, setting IOU<sub>T</sub> = 0 will decrease the number of pages of output appreciably.

If the user enters IOU<sub>T</sub> = 2, the long form of the printout is used. The long form of the output contains everything that the standard form of the output contains plus additional information. Usually, however, the additional items printed are of no interest to the user. Also, the long form of the output adds appreciably to the number of pages of printed output, especially for a steady state analysis run.

The following items are added to the standard printout for transient analysis runs when the long form of the output is requested: more detailed information about the flexible bladed disk(s) (if present) (including printout of the mass matrix for the rotor that includes the flexible bladed

disks and for the flexible bladed disks), the (mode,point,direction) trios that contribute to the generalized forces, the generalized forces, generalized velocities, generalized displacements, generalized accelerations, and a listing of at least part of the output plot file (lists the first 11 time steps on the plot file for a transient analysis run). The listing of the output plot file would be helpful to someone writing a postprocessor program to plot the TETRA results.

The following items are added to the standard printout for steady state analysis runs when the long form of the output is requested: more detailed information about the flexible bladed disk(s) (if present) (including the terms of the global mass and global velocity matrices due to the flexible bladed disks), the (mode,point,direction) trios that contribute to the generalized forces, the difference in the mode shapes at the I and J ends for each rub (type 3) element, a listing of at least part of the output plot file (lists the first 11 forcing frequencies on the plot file for a steady state analysis run), several matrices that are calculated for each of the linear (type 1, 2, 4, or 5) physical connecting element and each gyroscopic element, the global matrices, various other matrices and vectors, the generalized forces, the generalized displacements, the parameters used in the iterating equations, the sine and cosine components of the physical displacements, the sine and cosine components of the physical velocities, the sine and cosine components of the modal forces, the sine and cosine components of the physical connecting element forces, and the sine and cosine components of the gyroscopic element forces. Note that while the standard form of the output omits the sine and cosine components of the physical displacements, physical velocities, modal forces, physical connecting element forces, and gyroscopic element forces, the standard form of the output does print out the magnitudes and phase angles for these quantities. The magnitudes and phase angles (not the sine cosine components) are also what gets written to the plot file (usually the user is just interested in plotting the magnitudes).

Two other input variables also have significant effect on the length of the printed output. These are the print multiples (IPRMUL for a transient analysis and JPRMUL for a steady state analysis). These variables are entered on input sheet I-1. See the discussion of the type I-1 input sheet for details.

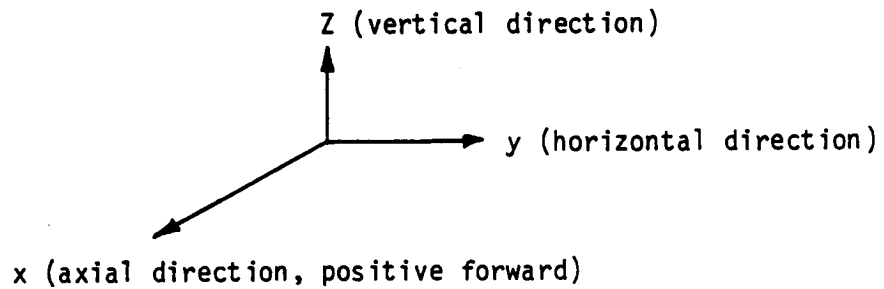
### 2.3.2 Physical Points Not Located On Modal Subsystems (Type B Input Sheet)

This input sheet defines the boundary conditions of the problem. The points which the user desires to ground are defined here. If the code is specified as fixed (code = 1), the point is fixed in all 6 directions. If the code is specified as free (code = 0), the point is free to move all 6 directions. The points defined on this sheet may not be located on the modal subsystems but can only be located at ground or at the junctions of link and engine support elements.



TETRA

PHYSICAL POINTS NOT LOCATED ON MODAL SUBSYSTEMS



PP=

POINT NUMBER NOT ON MODAL SUBSYSTEM *	CODE 0=FREE 1=FIXED	COORDINATES RELATIVE TO GLOBAL SYSTEM		
		X	Y	Z
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____

\* These points include ground points and points between link and engine support elements. It will be noted that points on modal subsystems cannot be attached directly to ground, but can be attached to ground through physical connecting elements.

### 2.3.3 Modal Subsystem Input (Type C-1 through C-16)

Modal subsystem input is accomplished using the LIST2 namelist section. One LIST2 namelist section is required for each modal subsystem. The number of modal subsystems is counted by the program automatically.

First, a discussion of the input that is common for all the modal subsystems. The user may input a title (this is optional) for each subsystem using the TITLE input variable. Up to 60 characters may be enclosed within the apostrophes for the title.

The subsystem number (ISUB input variable) is required for all modal subsystems. The subsystem number must be an integer between 1 and 13. The subsystems are not arbitrary - each subsystem number represents a different type of modal subsystem with differing degrees of freedom (see table 2-1). The user chooses which of the modal subsystems to use, and can arrange these modal subsystems in any order. The user must have at least one modal subsystem.

Note that subsystems 1 through 3 represent "rotor 1" and subsystems 4 through 6 represent "rotor 2" (see table 2-1). Of course, the user's model may have only one rotor, in which case either rotor 1 (subsystems 1 through 3) or rotor 2 (subsystems 4 through 6) could be used. If the engine is a dual spool engine, however, the user might want to use both "rotor 1" and "rotor 2" in his model. In this instance, "rotor 1" might represent, say, the low pressure system (fan, low pressure compressor, shaft, and low pressure turbine), and "rotor 2" might represent the high pressure system (high pressure compressor, shaft, and high pressure turbine). Or vice-versa - the choice is up to the user. The user has a great deal of flexibility and with these subsystems can model virtually any jet engine configuration.

Next, the input variables XREF, YREF, ZREF, and PTS are used for almost all of the subsystems. Specifically, these variables are used for the rotor, case, and pylon subsystems (subsystem numbers 1 through 11), but are not used for the flexible bladed disk subsystems (subsystems 12 and 13). If the subsystem reference point does not coincide with the global origin, the user must input the coordinates of the subsystem reference point (XREF, YREF, AND ZREF) input variables. Then, the user enters the points of the subsystems and their coordinates with respect to the subsystem reference point via the PTS input variable. Each subsystem must have at least one point. The current upper limits are 10 points for the rotor subsystems (subsystem numbers 1 through 6), 20 points for the case subsystems (subsystem numbers 7 through 10), and 10 points for the pylon subsystem (subsystem number 11).

The following sections detail the input that is specific to a particular type of subsystem.

Table 2-1. Modal Subsystems.

<u>SUBSYSTEM NUMBER</u>	<u>DESCRIPTION</u>	<u>NUMBER OF DEGREES OF FREEDOM</u>
1	Rotor 1 Vertical Plane	2
2	Rotor 1 Horizontal Plane	2
3	Rotor 1 Rigid Body	5
4	Rotor 2 Vertical Plane	2
5	Rotor 2 Horizontal Plane	2
6	Rotor 2 Rigid Body	5
7	Case Vertical Plane	2
8	Case Horizontal Plane	2
9	Case Rigid Body	6
10	Case Torsional	1
11	Pylon	3
12	Flexible Bladed Disk Number 1	2
13	Flexible Bladed Disk Number 2	2

### 2.3.3.1 Vertical and Horizontal Plane Subsystems (Type C-1 through C-3)

Input sheets C-1 through C-3 apply for the vertical and horizontal plane subsystems, that is, subsystem 1 (rotor 1 vertical plane), subsystem 2 (rotor 1 horizontal plane), subsystem 4 (rotor 2 vertical plane), subsystem 5 (rotor 2 horizontal plane), subsystem 7 (case vertical plane), and subsystem 8 (case horizontal plane). The first input variables for the vertical and horizontal plane subsystems are TITLE, ISUB, XREF, YREF, ZREF, and PTS. These variables were discussed in section 2.3.3.

Next, the frequency  $\omega$ , potential energy PE, Q-factor  $Q_f$ , and mode type code are entered for each mode using the XMODES input variable. There must be at least one mode entered. The current upper limit is 15 modes for the rotor subsystems (subsystem numbers 1 through 6), 30 modes for the case subsystems (subsystem numbers 7 through 10), and 30 modes for the pylon subsystem (subsystem number 11).

The input frequency  $\omega$  and input potential energy PE are used to calculate the generalized mass ZM for the mode as follows:

$$ZM = \frac{2PE}{\omega^2}$$

The Q-factor  $Q_f$  determines the modal damping for the mode (provided that the mode type code equals 0) and is defined as follows:

$$Q_f = \frac{1}{2(C/C_c)}$$

where C is the damping coefficient and  $C_c$  is the critical damping coefficient. If the user wishes to neglect damping,  $Q_f=0$  should be inputted, which is a signal to the computer to neglect damping.

The input mode type code must be 0 (flexible) or 1 (rigid body). The code 1 is used for "soft spring" rigid body modes, which are used to approximate the true rigid body modes. If this code equals 0 (flexible), the modal stiffness ZK and the modal damping ZC are calculated as follows:

$$ZK = 2PE \quad ; \quad ZC = \frac{ZK}{\omega Q_f}$$

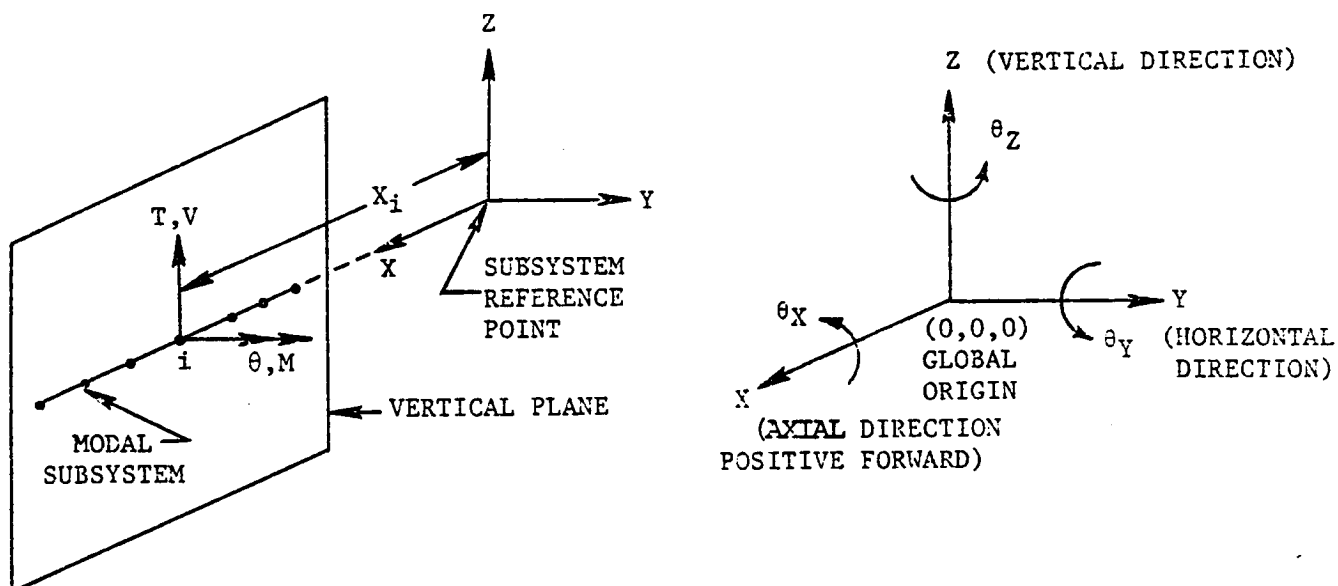
However, if the mode type code equals 1 (rigid body), then the modal stiffness and modal damping are both set to 0, regardless of what was input for PE and  $Q_f$ .

Finally, the mode shape input (translation, slope, shear, and moment) must be entered using the VH input variable. Mode shapes must be entered for every mode that was included in the XMODES input.

MODAL SUBSYSTEM INPUT FOR VERTICAL AND  
HORIZONTAL PLANE SUBSYSTEMS

THIS INPUT SHEET APPLIES FOR THE FOLLOWING MODAL SUBSYSTEMS:

- NUMBER 1 (ROTOR 1 VERTICAL PLANE)
- NUMBER 2 (ROTOR 1 HORIZONTAL PLANE)
- NUMBER 4 (ROTOR 2 VERTICAL PLANE)
- NUMBER 5 (ROTOR 2 HORIZONTAL PLANE)
- NUMBER 7 (CASE VERTICAL PLANE)
- NUMBER 8 (CASE HORIZONTAL PLANE)

NOTE - THE PHYSICAL MODEL (MODAL SOURCE) FOR THE SUBSYSTEM IS ALWAYS A  
VERTICAL PLANE MODEL.

▽  
\$

\$LIST2

TITLE= \_\_\_\_\_,

ISUB=\_\_\_\_\_, SUBSYSTEM NUMBER (1,2,4,5,7,or 8)

ENTER THE COORDINATES (INCHES) OF THE SUBSYSTEM REFERENCE POINT:

XREF=\_\_\_\_\_, X COORDINATE (0 ASSUMED)

YREF=\_\_\_\_\_, Y COORDINATE (0 ASSUMED)

ZREF=\_\_\_\_\_, Z COORDINATE (0 ASSUMED)

Maximum of 60 characters enclosed within apostrophes for the title.

TETRA

MODAL SUBSYSTEM INPUT FOR VERTICAL AND  
 HORIZONTAL PLANE SUBSYSTEMS (CONTINUED)

ENTER THE COORDINATES OF THE TETRA POINTS. MAXIMUM OF 10  
 POINTS FOR SUBSYSTEMS 1,2,4,AND 5. MAXIMUM OF 20 POINTS FOR  
 SUBSYSTEMS 7 AND 8. Y AND Z COORDINATES NORMALLY 0.  
 n=NUMBER OF POINTS IN THE SUBSYSTEM.

TETRA POINT NUMBER	COORDINATES (INCHES) WITH RESPECT TO SUBSYSTEM REFERENCE POINT		
	X	Y	Z

▽

PTS =

LOCAL POINT NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	.				
	.				
	n	_____	_____	_____	_____

ENTER THE VALUES BELOW FOR EACH MODE. MAXIMUM OF  
 30 MODES.  
 N=NUMBER OF SUBSYSTEM MODES.

FREQUENCY (RPM)	POTENTIAL ENERGY	Q-FACTOR	MODE TYPE 0=FLEXIBLE 1=RIGID BODY
--------------------	---------------------	----------	---

▽

XMODES =

LOCAL MODE NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	.				
	.				
	N	_____	_____	_____	_____

TETRAMODAL SUBSYSTEM INPUT FOR VERTICAL AND  
HORIZONTAL PLANE SUBSYSTEMS (CONTINUED)

ENTER THE REQUIRED MODE SHAPES BELOW.  
 n=NUMBER OF POINTS IN THE SUBSYSTEM.  
 N=NUMBER OF SUBSYSTEM MODES.

TRANSLATION T INCHES	SLOPE $\theta$ RADIAN	SHEAR V POUNDS	MOMENT M IN-LB
-------------------------	--------------------------	-------------------	-------------------



VH(1,1,1)=

LOCAL MODE 1 LOCAL POINT NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	.				

VH(1,1,2)=

LOCAL MODE 2 LOCAL POINT NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	.				

VH(1,1,N)=

LOCAL MODE N LOCAL POINT NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	.				

### 2.3.3.2 Rigid Body Subsystems (Type C-4 through C-6)

Input sheets C-4 through C-6 apply for the rigid body subsystems, that is, subsystem 3 (rotor 1 rigid body), subsystem 6 (rotor 2 rigid body), and subsystem 9 (case rigid body). The first input variables for the rigid body subsystems are TITLE, ISUB, XREF, YREF, ZREF, and PTS. These input variables were discussed in section 2.3.3.

Next, input variables XCG, YCG, and ZCG are used to input the coordinates of the subsystem center of gravity with respect to the subsystem reference point (0 is the assumed value for these quantities if not input).

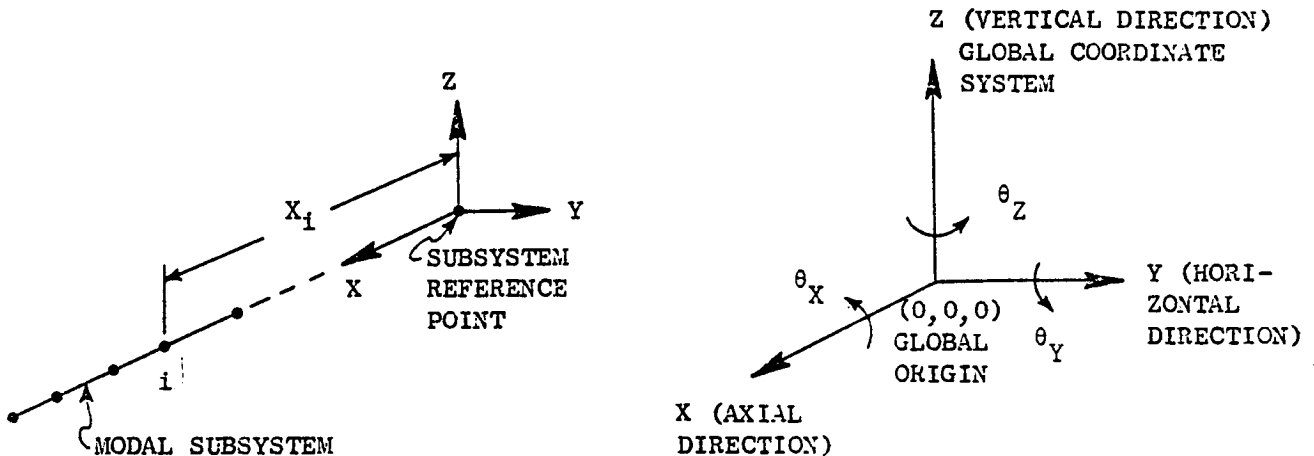
In addition, the physical weight properties (WX, WY, WZ, WMIX, WMIY, and WMIZ) should be input, but only for those directions for which rigid body motion is to be considered. Generalized coordinates are assigned for each direction for which the physical weight property is non-zero, but are not assigned if the physical weight property was set to zero or omitted. Since the  $\theta_x$  direction is always neglected for subsystems 3 and 6, WMIX should be set to zero or omitted for subsystems 3 and 6. At least one of the physical weight properties must be non-zero.

Note that there are two ways of accounting for the rigid body modes, that is, by entering the "soft spring" rigid body modes with the vertical and horizontal plane subsystem input (setting the mode type code to 1) (input sheets C-1 through C-3), or entering the rigid body modes with the rigid body subsystem input (input sheets C-4 through C-6). The user is cautioned not to enter these modes both places to avoid doubling the effect of these modes. Both ways of modeling the rigid body motion give equivalent results, as confirmed by test cases. However, it has been found that TETRA runs faster (and thus saves money) if the rigid body modes are included in the vertical and horizontal plane subsystems rather than the rigid body subsystems. Thus, it is recommended that the user use the vertical and horizontal plane subsystems in lieu of the rigid body subsystems for the rigid body modes. An exception to this guideline is if the user needs to account for rigid body motion in the X (axial) or  $\theta_x$  (torsional) direction, in which case the rigid body subsystem would be needed since the vertical and horizontal plane subsystems do not consider these directions.



MODAL SUBSYSTEM INPUT FOR RIGID BODY SUBSYSTEMS

THIS INPUT SHEET APPLIES FOR MODAL SUBSYSTEM 3 (ROTOR 1 RIGID BODY), MODAL SUBSYSTEM 6 (ROTOR 2 RIGID BODY), AND MODAL SUBSYSTEM 9 (CASE RIGID BODY).



NOTE: MOTION IN THE  $\theta_X$  DIRECTION IS NOT CONSIDERED FOR THE ROTOR SUBSYSTEMS (3 AND 6). MOTION IN THIS DIRECTION CAN BE CONSIDERED FOR THE CASE SUBSYSTEM (9), HOWEVER.

2

\$

\$LIST2

TITLE= ' \_\_\_\_\_ ',

ISUB= \_\_\_\_\_, SUBSYSTEM NUMBER (3, 6, OR 9)

ENTER THE COORDINATES (INCHES) OF THE SUBSYSTEM REFERENCE POINT:

XREF= \_\_\_\_\_, X COORDINATE (0 ASSUMED)

YREF= \_\_\_\_\_, Y COORDINATE (0 ASSUMED)

ZREF= \_\_\_\_\_, Z COORDINATE (0 ASSUMED)

Maximum of 60 characters enclosed within apostrophes for the title.

TETRA

PAGE \_\_\_\_ OF \_\_\_\_  
NAMELIST  
TYPE C-5

MODAL SUBSYSTEM INPUT FOR RIGID BODY SUBSYSTEMS (CONTINUED)

ENTER COORDINATES OF TETRA POINTS. MAXIMUM OF 10 POINTS FOR  
SUBSYSTEMS 3 AND 6 AND MAXIMUM OF 20 POINTS FOR SUBSYSTEM 9.  
Y AND Z COORDINATES NORMALLY 0.  
n= NUMBER OF SUBSYSTEM POINTS.

TETRA POINT NUMBER	COORDINATES (INCHES) WITH RESPECT TO SUBSYSTEM REFERENCE POINT		
	X	Y	Z

PTS =

LOCAL POINT NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.	.			
	.	.			
	.	.			
	.	.			
	.	.			
	.	.			
	n	_____	_____	_____	_____

ENTER THE SUBSYSTEM CENTER OF GRAVITY COORDINATES (INCHES) WITH  
RESPECT TO SUBSYSTEM REFERENCE POINT:

XCG= \_\_\_\_\_ X COORDINATE (0 ASSUMED)  
YCG= \_\_\_\_\_ Y COORDINATE (0 ASSUMED)  
ZCG= \_\_\_\_\_ Z COORDINATE (0 ASSUMED)

MODAL SUBSYSTEM INPUT FOR RIGID BODY SUBSYSTEMS (CONTINUED)

ENTER THE SUBSYSTEM PHYSICAL WEIGHT PROPERTIES FOR EACH OF THE SIX DIRECTIONS. IF MOTION IN A PARTICULAR DIRECTION IS TO BE NEGLECTED, ENTER ZERO FOR THE PHYSICAL WEIGHT PROPERTY IN THAT DIRECTION.

(SINCE THE 6X DIRECTION IS ALWAYS NEGLECTED FOR SUBSYSTEMS 3 AND 6, WMIX SHOULD ALWAYS BE SET TO ZERO OR OMITTED FOR SUBSYSTEMS 3 AND 6). AT LEAST ONE OF THE PHYSICAL WEIGHTS MUST BE NON-ZERO.

PHYSICAL WEIGHT PROPERTY	DIRECTION	X
		Y
		Z
		$\theta_X$
		$\theta_Y$
		$\theta_Z$

WX = \_\_\_\_\_, WEIGHT, POUNDS (0 ASSUMED)  
 WY = \_\_\_\_\_, WEIGHT, POUNDS (0 ASSUMED)  
 WZ = \_\_\_\_\_, WEIGHT, POUNDS (0 ASSUMED)  
 WMIX = \_\_\_\_\_, POLAR MOMENT OF INERTIA, LB-IN<sup>2</sup> (0 ASSUMED)  
 WMIY = \_\_\_\_\_, MOMENT OF INERTIA ABOUT Y AXIS, LB-IN<sup>2</sup> (0 ASSUMED)  
 WMIZ = \_\_\_\_\_, MOMENT OF INERTIA ABOUT Z AXIS, LB-IN<sup>2</sup> (0 ASSUMED)

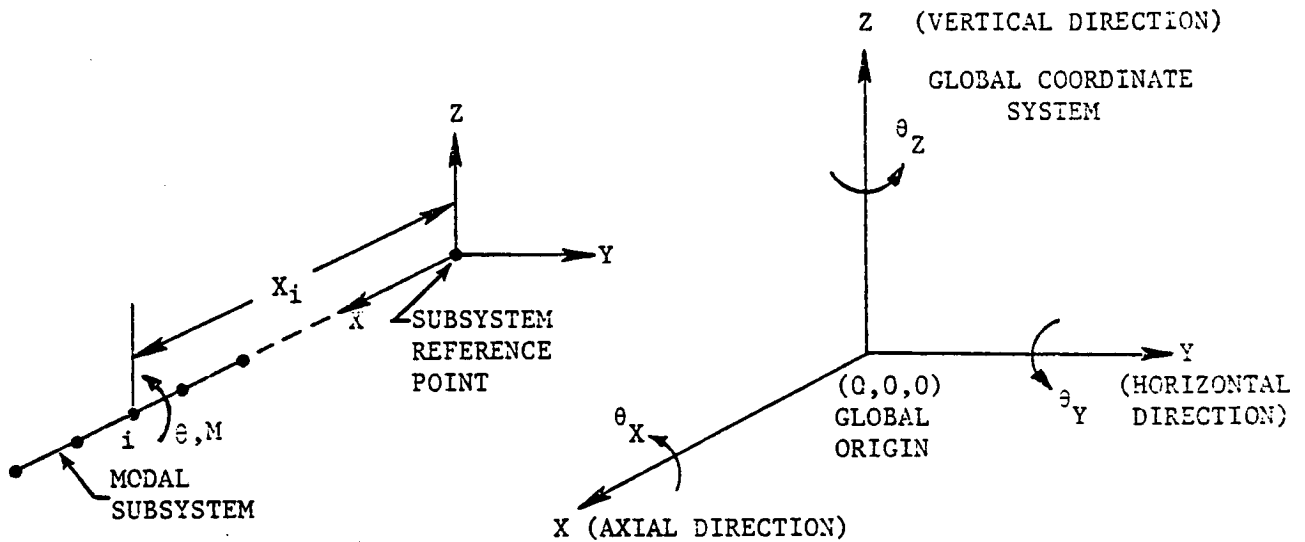
#### 2.3.3.3 Case Torsional Subsystem (Type C-7 through C-9)

Input sheets C-7 through C-9 are used for the case torsional subsystem, which is subsystem 10. The first input variables for this subsystem are TITLE, ISUB, XREF, YREF, ZREF, and PTS. These input variables were discussed in section 2.3.3. Next comes input variable XMODES, which was discussed in section 2.3.3.1.

The final input variable for the torsional subsystem is variable TOR, which is used to enter the mode shapes (twist and moment). Mode shapes must be entered for every mode that was included in the XMODES input. There must be at least one mode. The current upper limit for the case subsystems is 30 modes.

TETRA

MODAL SUBSYSTEM INPUT FOR SUBSYSTEM 10  
(CASE TORSIONAL SUBSYSTEM)



▽  
 \$  
 \$LIST2  
 TITLE= '\_\_\_\_\_  
 ISUB=10,

ENTER THE COORDINATES (INCHES) OF THE SUBSYSTEM REFERENCE POINT:

XREF=\_\_\_\_\_, X COORDINATE (0 ASSUMED)

YREF=\_\_\_\_\_, Y COORDINATE (0 ASSUMED)

ZREF=\_\_\_\_\_, Z COORDINATE (0 ASSUMED)

Maximum of 60 characters enclosed within apostrophes for the title.

TETRA

MODAL SUBSYSTEM INPUT FOR SUBSYSTEM 10  
 (CASE TORSIONAL SUBSYSTEM) (CONTINUED)

ENTER THE COORDINATES OF THE TETRA POINTS. MAXIMUM OF 20 POINTS.  
 X AND Y COORDINATES NORMALLY 0  
 n=NUMBER OF POINTS IN THE SUBSYSTEM

TETRA POINT NUMBER	COORDINATES (INCHES) WITH RESPECT TO SUBSYSTEM REFERENCE POINT		
	X	Y	Z

2

PTS =

LOCAL POINT NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	n	_____	_____	_____	_____

ENTER THE VALUES BELOW FOR EACH MODE. MAXIMUM OF 30 MODES.  
 N=NUMBER OF SUBSYSTEM MODES.

FREQUENCY (RPM)	POTENTIAL ENERGY	Q-FACTOR	MODE TYPE 0=FLEXIBLE 1=RIGID BODY
--------------------	---------------------	----------	---

2

XMODES =

LOCAL MODE NUMBER	1	_____	_____	_____	_____
	2	_____	_____	_____	_____
	3	_____	_____	_____	_____
	.				
	.				
	.				
	.				
	.				
	.				
	N	_____	_____	_____	_____

TETRA

MODAL SUBSYSTEM INPUT FOR SUBSYSTEM 10  
(CASE TORSIONAL SUBSYSTEM) (CONTINUED)

ENTER THE REQUIRED MODE SHAPES BELOW.  
 n=NUMBER OF POINTS IN THE SUBSYSTEM.  
 N=NUMBER OF SUBSYSTEM MODES.

TWIST $\theta$	MOMENT
RADIANS	IN-LB

2/

LOCAL MODE 1	LOCAL POINT NUMBER
	1
	2
	3
	.
	.
	.
	.
	.
	n

TOR(1,1,1)=

\_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_,  
 .  
 .  
 .  
 .  
 .  
 \_\_\_\_\_, \_\_\_\_\_,

LOCAL MODE 2	LOCAL POINT NUMBER
	1
	2
	3
	.
	.
	.
	.
	.
	n

TOR(1,1,2)=

\_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_,  
 .  
 .  
 .  
 .  
 .  
 .  
 \_\_\_\_\_, \_\_\_\_\_,  
 .  
 .  
 .  
 .

LOCAL MODE N	LOCAL POINT NUMBER
	1
	2
	3
	.
	.
	.
	.
	.
	.
	n

TOR(1,1,N)=

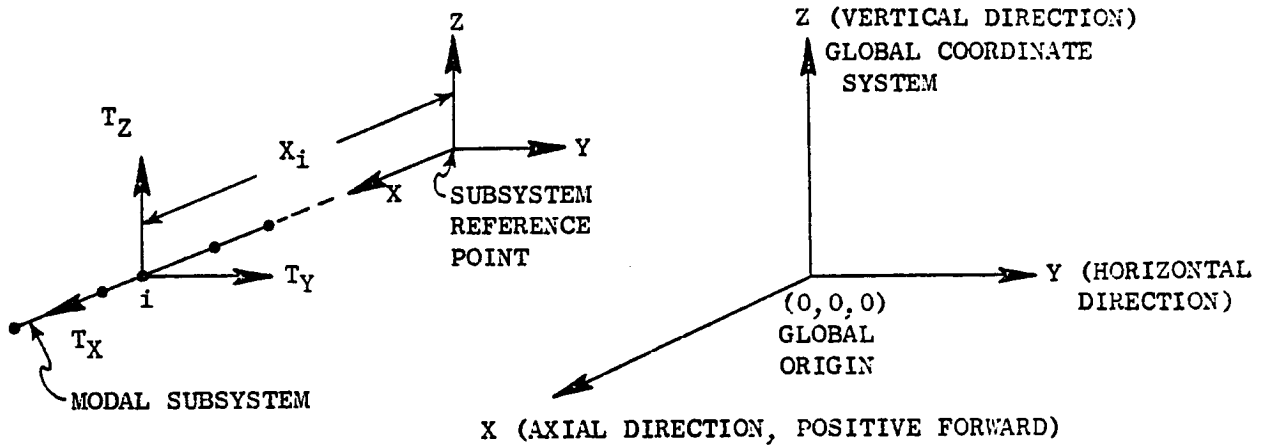
\_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_,  
 .  
 .  
 .  
 .  
 .  
 .  
 \_\_\_\_\_, \_\_\_\_\_,

#### 2.3.3.4 Pylon Subsystem (Type C-10 through C-12)

Input sheets C-10 through C-12 are used for the pylon subsystem, which is subsystem 11. The first input variables for this subsystem are TITLE, ISUB, XREF, YREF, ZREF, and PTS. These input variables were discussed in section 2.3.3. Next comes input variable XMODES, which was discussed in section 2.3.3.1.

The final input variable for the pylon subsystem is variable PYL, which is used to enter the mode shapes (translation in the X, Y, and Z directions). Mode shapes must be entered for every mode that was included in the XMODES input. There must be at least one mode. The current upper limit for the pylon subsystem is 30 modes.



MODAL SUBSYSTEM INPUT FOR SUBSYSTEM 11  
PYLON SUBSYSTEM

2

\$

\$LIST2

TITLE= ' \_\_\_\_\_ ' ,

ISUB=11,

ENTER THE COORDINATES (INCHES) OF THE SUBSYSTEM REFERENCE POINT:

XREF= \_\_\_\_\_, X COORDINATE (0 ASSUMED)

YREF= \_\_\_\_\_, Y COORDINATE (0 ASSUMED)

ZREF= \_\_\_\_\_, Z COORDINATE (0 ASSUMED)

Maximum of 60 characters enclosed within apostrophes for the title.

TETRA

MODAL SUBSYSTEM INPUT FOR SUBSYSTEM 11  
(PYLON SUBSYSTEM) (CONTINUED)

ENTER THE COORDINATES OF THE TETRA POINTS. MAXIMUM OF 10 POINTS.  
 n=NUMBER OF POINTS IN THE SUBSYSTEM

TETRA POINT NUMBER	COORDINATES (INCHES) WITH RESPECT TO SUBSYSTEM REFERENCE POINT		
	X	Y	Z

2

PTS =

LOCAL POINT NUMBER	1
	2
	3
	.
	.
	.
	.
	.
	.
	N

_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
_____	_____	_____	_____

ENTER THE VALUES BELOW FOR EACH MODE. MAXIMUM OF 30 MODES.  
 N=NUMBER OF SUBSYSTEM MODES.

FREQUENCY (RPM)	POTENTIAL ENERGY	Q-FACTOR	MODE TYPE 0=FLEXIBLE 1=RIGID BODY
--------------------	---------------------	----------	---

2

XMODES =

LOCAL MODE NUMBER	1
	2
	3
	.
	.
	.
	.
	.
	.
	N

_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
.	.	.	.
_____	_____	_____	_____

TETRA

MODAL SUBSYSTEM INPUT FOR SUBSYSTEM 11  
PYLON SUBSYSTEM (CONTINUED)

ENTER THE REQUIRED MODE SHAPES BELOW.  
 n= NUMBER OF POINTS IN THE SUBSYSTEM.  
 N= NUMBER OF SUBSYSTEM MODES.

TRANSLATION $T_x$ INCHES	TRANSLATION $T_y$ INCHES	TRANSLATION $T_z$ INCHES
--------------------------------	--------------------------------	--------------------------------

2

PYL(1,1,1)=

LOCAL MODE 1 LOCAL POINT NUMBER	1	_____	_____	_____
	2	_____	_____	_____
	3	_____	_____	_____
	.	.	.	.
	.	.	.	.
	.	.	.	.
	.	.	.	.
	n	_____	_____	_____

PYL(1,1,2)=

LOCAL MODE 2 LOCAL POINT NUMBER	1	_____	_____	_____
	2	_____	_____	_____
	3	_____	_____	_____
	.	.	.	.
	.	.	.	.
	.	.	.	.
	.	.	.	.
	n	_____	_____	_____

PYL(1,1,N)=

LOCAL MODE N LOCAL POINT NUMBER	1	_____	_____	_____
	2	_____	_____	_____
	3	_____	_____	_____
	.	.	.	.
	.	.	.	.
	.	.	.	.
	.	.	.	.
	n	_____	_____	_____

#### 2.3.3.5 Flexible Bladed Disk Subsystems (Type C-13 through C-16)

Input sheets C-13 through C-16 are used for the flexible bladed disk subsystems, that is, subsystem 12 (flexible bladed disk number 1) and subsystem 13 (flexible bladed disk number 2). For more details pertaining to the flexible bladed disks, see reference 2.

A TETRA run can have 0, 1, or 2 flexible bladed disks. If two flexible bladed disks are present, they must be located on the same rotor. If the flexible bladed disks are located on rotor 1, then the rigid body modes for rotor 1 must be included in subsystems 1 and 2, and subsystem 3 must be omitted. If the flexible bladed disks are on rotor 2, then the rigid body modes for rotor 2 must be included in subsystems 4 and 5, and subsystem 6 must be omitted. Both the rotor vertical and horizontal plane subsystems should be included for the rotor which includes the flexible bladed disk(s) to account for coupling between the generalized coordinates. The physical weight and rigid moment of inertia properties of the flexible bladed disk(s) must be included in the applicable rotor vertical and horizontal plane subsystems. To account for gyroscopic effects, the polar moment of inertia of the flexible bladed disk must be entered on input sheet N at the flexible bladed disk center of gravity point. Rotor speed (input sheets J-1 and J-2) must be included for the rotor on which the flexible bladed disk(s) are located.

The first input variables are TITLE and ISUB. These input variables were discussed in section 2.3.3. The next input variable is ICG, the TETRA point number for the center of gravity of the flexible bladed disk. This point must be included in the applicable rotor vertical and horizontal plane input.

Next, several flexible bladed disk properties are input, namely WTF (one diameter modal weight), QFAC (the modal Q-factor), WF (static frequency), XMU (modal tangential shear coefficient), SV (modal moment coefficient), and BETA (rpm-beta factor table). If damping is not desired, the modal Q-factor QFAC should be set to 0 or omitted, which is a signal to the program to neglect damping. The rpm-beta factor table must include at least 2 and a maximum of 10 lines, with the table entries in order of increasing rpm.

Following this comes the input for the local points on the flexible bladed disk. This includes input variables FBDPTS (the radius and polar angle for each local point), FBDMS (the static mode shapes, that is, tangential translation and axial translation at the local points), and FBDS (the three modal stress components at the local points). Displacements and stresses are output for each of the local points. There may be zero local points if the user is not interested in flexible bladed disk displacements and stresses. The maximum number of local points permitted is 200 for each flexible bladed disk.

ORIGINAL PAGE IS  
OF POOR QUALITY

Page \_\_\_\_\_ of \_\_\_\_\_  
NAMELIST  
Type C-13

MODAL SUBSYSTEM INPUT FOR FLEXIBLE BLADED DISK SUBSYSTEMS

Input sheets C-13 through C-16 apply for Subsystem 12 (Flexible Bladed Disk Number 1) and Subsystem 13 (Flexible Bladed Disk Number 2).

If both FBD Number 1 and FBD Number 2 are present, they must be located on the same rotor. If the FBD(s) are on Rotor 1, then the rigid body modes for Rotor 1 must be included in modal Subsystems 1 and 2, and modal Subsystem 3 must be omitted. If the FBD(s) are on Rotor 2, then the rigid body modes for Rotor 2 must be included in Subsystems 4 and 5, and Subsystem 6 must be omitted. Both the rotor vertical plane and horizontal plane subsystems should be included for the rotor which includes the FBD(s) to account for coupling between the generalized coordinates. The physical weight and rigid moment of inertia properties of the FBD(s) must be included in the applicable rotor vertical and horizontal plane subsystems. Rotor speed input (Input Sheets J-1 and J-2) must be included for the rotor on which the FBD(s) are located.

2

\$END

\$LIST2

TITLE=' \_\_\_\_\_ '

ISUB= \_\_\_\_\_, SUBSYSTEM NUMBER (12 or 13)

ICG= \_\_\_\_\_, TETRA Point number for the FBD center of gravity (This point must be included in the applicable rotor vertical and horizontal plane subsystem input)

Maximum of 60 characters enclosed within apostrophes for the title.

MODAL SUBSYSTEM INPUT FOR FLEXIBLE BLADED DISK SUBSYSTEMS

2

WTF= \_\_\_\_\_, one-diameter modal weight  $WT_f$  (lb)  
QFAC= \_\_\_\_\_, modal Q-factor (omit or set to 0 if no damping  
desired)  
WF= \_\_\_\_\_, static frequency  $w_f$  (hertz)  
XMU= \_\_\_\_\_, modal tangential shear coefficient,  $M_u$  (lb)  
SV= \_\_\_\_\_, modal moment coefficient,  $S_v$  (in-lb)

Note: To account for gyroscopic effects for the FBD, the polar moment of inertia of the FBD must be input on Sheet N at the FBD C.G. point.

Input the following table for beta factors. Include at least 2 and a maximum of 10 lines. Entries must be in order of increasing rpm.

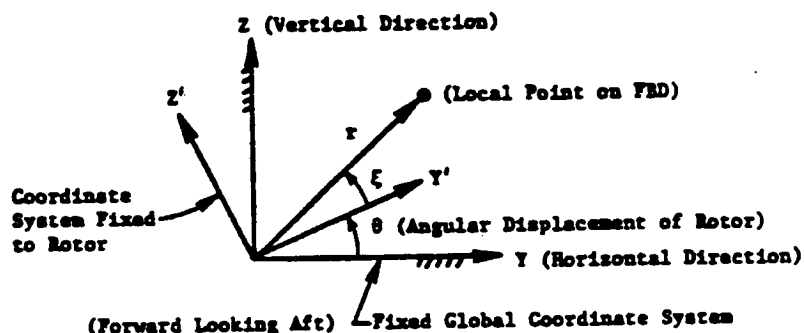
RPM	$\beta$ FACTOR
-----	-------------------

2

BETA=

\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,

MODAL SUBSYSTEM INPUT FOR FLEXIBLE BLADED DISK SUBSYSTEMS (Continued)



Enter the  $r$  and  $\xi$  coordinates for each of the subsystem points other than the center of gravity point. Maximum of 200 points for subsystem 12 and 200 points for subsystem 13 other than the center of gravity points.  $n$  = number of subsystem points other than the center of gravity point.

Radius	Polar Angle
$r$	$\xi$
inches	degrees

$$r \geq 0$$

$$0^\circ \leq \xi \leq 360^\circ$$

FBDPTS=

Local Point Number	1	_____	_____
	2	_____	_____
	3	_____	_____
	...	_____	_____
	...	_____	_____
	...	_____	_____
	...	_____	_____
	...	_____	_____
	...	_____	_____
	n	_____	_____

MODAL SUBSYSTEM INPUT FOR FLEXIBLE BLADED DISK SUBSYSTEMS (Concluded)

For each of the subsystem points other than the C.G. point enter the static (zero speed) mode shapes (assumed to be 0 if not entered):

FBD Tangential Translation u (inches)	FBD Axial Translation v (inches)
--	---

2

FBDMS =

Local Point Number	1	2	3	.	.	.	.	.	.	.	n
	_____	_____	_____	_____	_____	_____	_____	_____	_____	_____	_____

For each of the subsystem points other than the C.G. point enter the modal stress components (assumed to be 0 if not entered):

$\sigma_1$ psi	$\sigma_2$ psi	$\sigma_3$ psi
-------------------	-------------------	-------------------

2

FBDS =

Local Point Number	1	2	3	.	.	.	.	.	.	.	n
	_____	_____	_____	_____	_____	_____	_____	_____	_____	_____	_____



#### 2.3.4. Physical Connecting Element Input (Type D-1 through H-3)

TETRA 2 has capability for 6 types of physical connecting elements. These types are the general spring-damper element (type 1), the space link-damper element (type 2), the rub element (type 3), the engine support-links element (type 4), the uncoupled point spring-damper element (type 5), and the squeeze film damper element (type 6). All of these types may be used for either transient or steady state analysis runs with the exception of the squeeze film damper element, which may be used only for transient analysis runs.

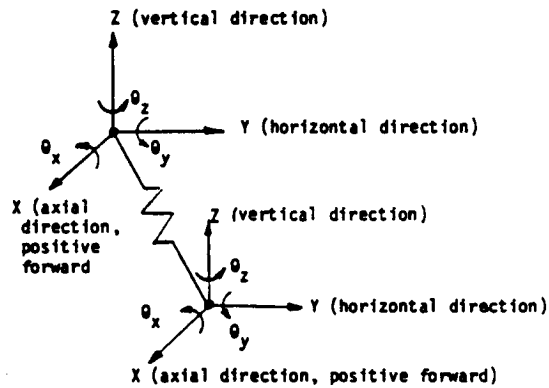
A new feature added to TETRA 2 is structural damping capability, which applies for the linear (type 1, 2, 4, and 5) physical connecting elements. This feature can be used either for transient or steady state analyses. As one can see from the input sheets, the input variables QELEM (q-factor) and QFREQ (selected frequency) may be used for any of the linear physical connecting elements. If both QELEM and QFREQ are entered, then damping is calculated based on the entered q-factor and entered selected frequency. This type of damping is non-structural, and was present in the earlier versions of TETRA. However, if QELEM is entered but not QFREQ, then damping is based on the entered Q-factor and either the independent rotor speed or the steady state forcing frequency, as determined by input variable ISF on the type A input sheet (see the discussion of the type A input sheet). This type of damping, which varies as the independent rotor speed or the steady state forcing frequency changes, is structural damping, and was not present in the earlier versions of TETRA.

#### 2.3.4.1 General Spring-Damper Element (Type D-1 and D-2)

This element is associated with two physical points located at arbitrary locations in global space. Each of these points is assigned six degrees of freedom, three translational displacements (or velocities), and three rotational displacements (or velocities).

A full complement of stiffness and damping coefficients can be input to allow the modeling of fully coupled load paths. The units of the stiffness matrix coefficients are: lb/in, lb/rad, in-lb/in, and in-lb/rad. The units of the damping matrix coefficients are lb-sec/in, lb-sec/rad, in-lb-sec/in, and in-lb-sec/rad.

Damping, if present, can be specified directly via damping matrix definition (non-structural damping), may be calculated from the input q-factor and selected frequency (non-structural damping), or may be calculated based on the input q-factor and either the independent rotor speed or the steady state forcing frequency as determined by input variable ISF on the type A input sheet (structural damping). See section 2.3.4 for further discussion of structural damping.

TYPE 1 PHYSICAL CONNECTING ELEMENT(GENERAL SPRING-DAMPER ELEMENT)

2

\$

\$LIST3

ITYPE=1,

ILEM=\_\_\_\_\_, element number

I-end                  J-end  
 point number point number

JT=\_\_\_\_\_, \_\_\_\_\_,

STIFFNESS MATRIX DEFINITION											
I-END						J-END					
GLOBAL DIRECTION DISPLACEMENT						GLOBAL DIRECTION DISPLACEMENT					
1	2	3	4	5	6	1	2	3	4	5	6
Z	$\theta_y$	Y	$\theta_z$	X	$\theta_x$	Z	$\theta_y$	Y	$\theta_z$	X	$\theta_x$

Global Dir. Forces

I-END	1	$F_z$
	2	$F_{\theta_y}$
	3	$F_y$
	4	$F_{\theta_z}$
	5	$F_x$
	6	$F_{\theta_x}$
J-END	1	$F_z$
	2	$F_{\theta_y}$
	3	$F_y$
	4	$F_{\theta_z}$
	5	$F_x$
	6	$F_{\theta_x}$

SPRING=

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

TYPE 1 PHYSICAL CONNECTING ELEMENT(GENERAL SPRING-DAMPER ELEMENT) (Continued)

2

1=damping

IDAMP= 0=no damping, (0 assumed)

If IDAMP=1, complete the input for one of the following two options:

Option 1: For damping based on Q-factor and selected frequency, enter the following (if QELEM is entered but not QFREQ, the frequency defined by the ISF q-factor variable on type A input sheet is used).

QELEM= \_\_\_\_\_,

frequency(hertz)

QFREQ= \_\_\_\_\_,

Option 2: For damping based on damping matrix definition, enter the following:

DAMPING MATRIX DEFINITION											
I-END						J-END					
GLOBAL DIRECTION VELOCITY						GLOBAL DIRECTION VELOCITY					
1	2	3	4	5	6	1	2	3	4	5	6
$\dot{z}$	$\dot{\theta}_y$	$\dot{y}$	$\dot{\theta}_z$	$\dot{x}$	$\dot{\theta}_x$	$\dot{z}$	$\dot{\theta}_y$	$\dot{y}$	$\dot{\theta}_z$	$\dot{x}$	$\dot{\theta}_x$

Global Dir. Forces		
I-END	1	$F_z$
	2	$F_{\theta y}$
	3	$F_y$
	4	$F_{\theta z}$
	5	$F_x$
	6	$F_{\theta x}$
J-END	1	$F_z$
	2	$F_{\theta y}$
	3	$F_y$
	4	$F_{\theta z}$
	5	$F_x$
	6	$F_{\theta x}$

DAMP =

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

#### 2.3.4.2 Space Link-Damper Element (Type E)

This element is used to model load paths which have no local moment capability. The load paths are associated with two physical points located at arbitrary locations in global space. Each of these points is assigned three translational degrees of freedom. The user inputs the cross sectional area and Young's modulus, and TETRA 2 uses this information along with the coordinates of the two connecting points to calculate the stiffness matrix.

Damping, if present, can be specified by the translational damping coefficient (non-structural damping), by the input q-factor and selected frequency (non-structural damping), and by the input q-factor and either the independent rotor speed or the steady state forcing frequency as determined by variable ISF on the type A input sheet (structural damping). See section 2.3.4 for more discussion about structural damping.

TYPE 2 PHYSICAL CONNECTING ELEMENT(SPACE LINK-DAMPER ELEMENT)

2

\$

\$LIST3

ITYPE=2,

element number

ILEM=\_\_\_\_\_,

I-end J-end

point number point number

JT=\_\_\_\_\_,

Area(in<sup>2</sup>)

TAREA=\_\_\_\_\_,

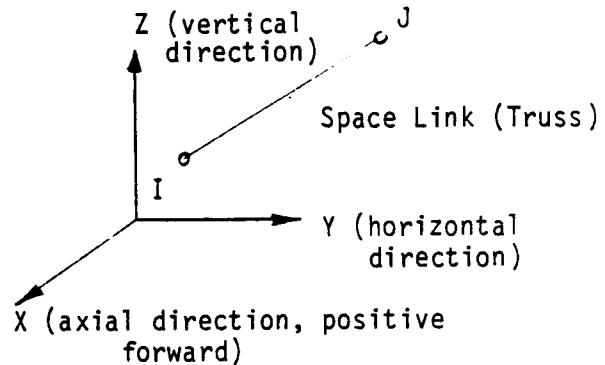
Young's Modulus (psi)

TYOUNG=\_\_\_\_\_,

1=damping

0=no damping

IDAMP=\_\_\_\_\_, (0 assumed)



If IDAMP=1, complete the input for one of the following two options:

Option 1: For damping based on Q-factor and selected frequency, enter the following (if QELEM is entered but not QFREQ, the frequency defined by the ISF variable on type A input sheet is used).

Q-factor

QELEM=\_\_\_\_\_,

frequency (hertz)

QFREQ=\_\_\_\_\_,

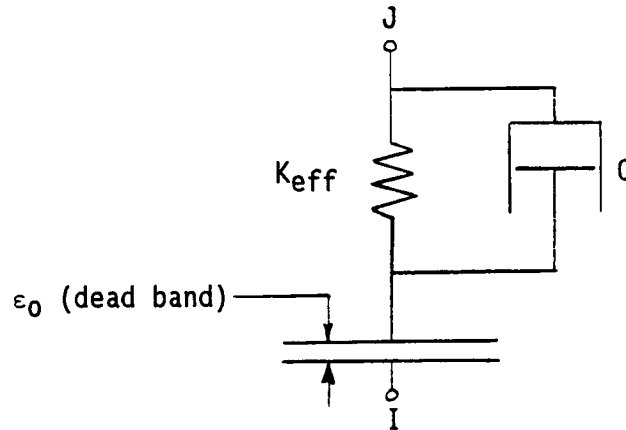
Option 2: For damping based on the translational damping coefficient c along the axis of the link element, enter the following:

C (lb-sec/in)

TDRATE=\_\_\_\_\_,

#### 2.3.4.3 Rub Element (Type F)

The rub element allows the mathematical modeling of the nonlinear tip rub, which includes dead band displacement (input variable DBAND) and a cubic nonlinear factor (input variable AK). The cubic nonlinear factor is a new feature that has been added to TETRA 2. This element can be used to model rotor to case rubs as well as rotor to rotor rubs. In the former case, the I end must be on the rotor and the J end on the case. In the latter case, the I end must be on the inner rotor and the J end on the outer rotor.

TYPE 3 PHYSICAL CONNECTING ELEMENT (RUB ELEMENT)

$$K_{eff} = SK \left(1 - \frac{\epsilon_0}{|\Delta|}\right) [1 + AK(|\Delta| - \epsilon_0)^2]$$

where  $|\Delta|$  = relative deflection between line of centers

2

\$  
\$LIST3  
ITYPE=3,  
ILEM=\_\_\_\_\_, element number

I end J end  
point number point number  
JT=\_\_\_\_\_, \_\_\_\_\_,

Note: If rotor-case rub, the I end must be on the rotor and the J end on the case. If rotor-rotor rub, the I end must be on the inner rotor and the J end on the outer rotor.

DBAND=\_\_\_\_\_, radial dead band  $\epsilon_0$  (mils) between rotor and case (if rotor-case rub) or between inner rotor and outer rotor (if rotor-rotor rub) (0 assumed)

SK=\_\_\_\_\_, linear factor (lb/in) for radial spring constant  $K_{eff}$  (becomes active on closure) (0 assumed)

AK=\_\_\_\_\_, nonlinear factor ( $\text{in}^{-2}$ ) for radial spring constant  $K_{eff}$  (becomes active on closure) (0 assumed)

CC=\_\_\_\_\_, damping coefficient C (lb-sec/in) (becomes active on closure) (0 assumed)



#### 2.3.4.4 Engine Support-Links Element (Type G-1 through G-3)

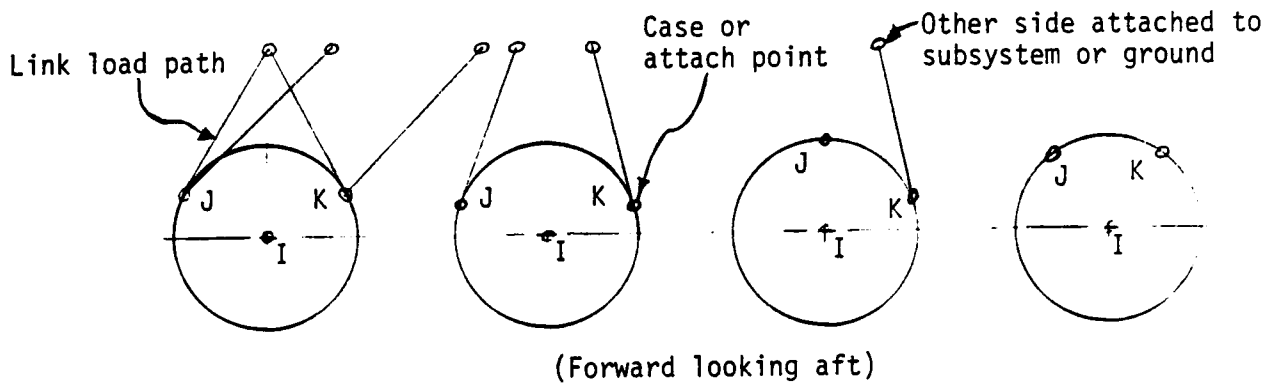
The engine support element is a multipoint, multidirection variable geometry element that provides the capability to model the complex load paths between the engine case and the pylon or ground and thus allows for the direct modeling of actual engine mount structures. In addition, this element couples the flexible and rigid body centerline modal subsystem that represents the engine case to the support links that comprise the mounting system through the case flexibility. The case flexibility is described by the case distortion flexibility rates SKV, SKH, and SKA for the vertical, horizontal, and axial directions respectively.

The flexibility rates are the reciprocals of the case spring rates that represent local distortion of the engine case under mount reaction loads. Multipliers, defined by the input variables AM, BM, CM, DM, EM, and FM, are used to proportion these flexibility rates as per the diagram shown on the type G-2 input sheet. If it is desired to cut a load path at a case attach point J or K in a given direction, then the applicable multiplier can be set equal to zero. The load paths between the engine case and the pylon or ground can be defined with up to 8 link elements. In defining these load paths, no more than 4 distinct points can be specified on the pylon or ground side. This means that more than a single link may be used to connect a case point to a pylon or ground point. The coordinates for the points that correspond to the J and K points on the case are defined on the Type B input sheet if these points are connected to links.

The damping, if present, is defined by the input q-factor and input selected frequency (non-structural damping), or is defined by the input q-factor and either the independent rotor speed or the steady state forcing frequency as determined by variable ISF on the type A input sheet (structural damping). For more discussion on structural damping, see section 2.3.4. For more discussion on the type 4 element, see reference 1.

TYPE 4 PHYSICAL CONNECTING ELEMENT (ENGINE SUPPORT-LINKS ELEMENT)

Some example configurations that can be modeled with the type 4 element:



2

\$  
\$LIST3  
ITYPE=4,  
ILEM= \_\_\_\_\_, element number

I-end  
point number  
(engine centerline)

J-end  
point number  
(Case point)

K-end  
point number  
(Case point)

JT= \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

Point I is attached to subsystem. Points J and K can connect to link load paths or subsystems or ground.

SKV= \_\_\_\_\_, Vertical direction case distortion  
flexibility rate (in/lb)

SKH= \_\_\_\_\_, Horizontal direction case distortion  
flexibility rate (in/lb)

SKA= \_\_\_\_\_, Axial direction case distortion  
flexibility rate (in/lb)

TYPE 4 PHYSICAL CONNECTING ELEMENTS (ENGINE SUPPORT-LINKS ELEMENT) (Continued)

Enter multipliers a, b, c, d, e, and f to proportion the spring rates:

2

AM= \_\_\_\_\_,  
 BM= \_\_\_\_\_,  
 CM= \_\_\_\_\_,  
 DM= \_\_\_\_\_,  
 EM= \_\_\_\_\_,  
 FM= \_\_\_\_\_,

Rate	POINT	
	J	K
Kv	a	c
KH	b	d
KA	e	f

$$\left. \begin{aligned} a + c &= 1.0 \\ b + d &= 1.0 \\ e + f &= 1.0 \end{aligned} \right\}$$

This restraint must be followed  
(zero values are permissible).

Input one line for each link load path (maximum of 8). If no link load paths, omit this input.

Case side point number (must correspond to case point J or K)	Other side point number (no more than 4 distinct point numbers)	Area (in <sup>2</sup> )	Young's Modulus psi
--	--	-------------------------	---------------------------

TLP(1,1)=

_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____

TETRA

Page \_\_\_ of \_\_\_  
NAMELIST  
Type G-3

TYPE 4 PHYSICAL CONNECTING ELEMENT (ENGINE SUPPORT-LINKS ELEMENT) (Continued)

Enter the following only if you want damping based on Q-factor and selected frequency. If no damping desired, omit this input. If QELEM is entered but not QFREQ, the frequency defined by the ISF variable on type A input sheet is used.

Q-factor

QELEM=\_\_\_\_\_.

frequency (hertz)

QFREQ=\_\_\_\_\_.

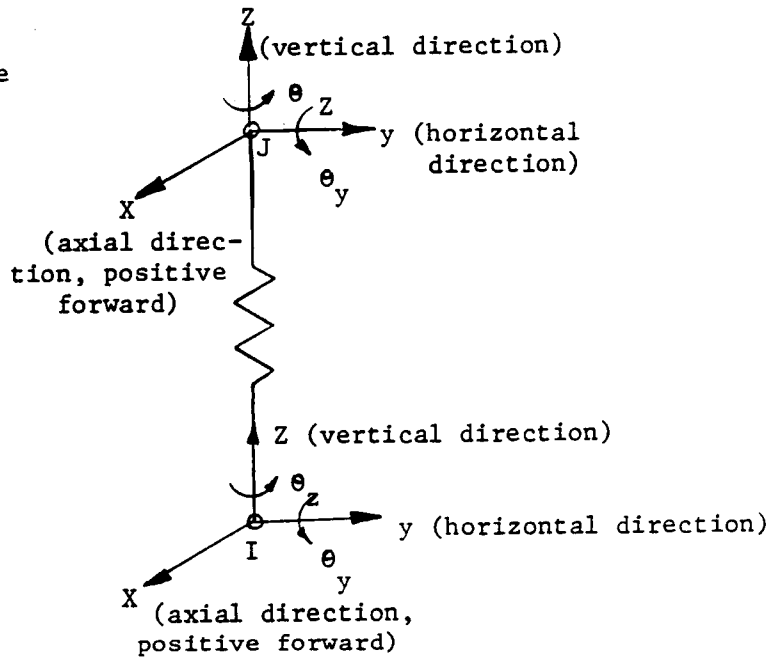
#### 2.3.4.5 Uncoupled Point Spring-Damper Element (Type H-1 and H-2)

This input allows the connection of two points with a set of uncoupled springs and dampers. This element is typically used to connect the centerlines of beam-like modal subsystems and provides load paths in three translational and two rotational directions. Because of the lack of load path coupling, good modeling practice infers that the points being connected by this element should be coincident in space.

Damping, if present, can be specified by the coefficient input (non-structural damping), by the input q-factor and selected frequency (non-structural damping), or by the input q-factor and either the independent rotor speed or the steady state forcing frequency as determined by variable ISF on the type A input sheet (structural damping). For more discussion about structural damping, see section 2.3.4.

TYPE 5 PHYSICAL CONNECTING ELEMENT  
(UNCOUPLED POINT SPRING-DAMPER ELEMENT)

I and J points are  
coincident.



2

\$

\$LIST3

ITYPE=5,

ILEM=\_\_\_\_\_, element number

I-end J-end  
point number point number

JT=\_\_\_\_\_, \_\_\_\_\_,

Enter the following spring constants:

$K_x$  (lb/in)

$K_y$  (lb/in)

$K_z$  (lb/in)

$K_{\theta_y}$  ( $\frac{\text{in-lb}}{\text{rad}}$ )

$K_{\theta_z}$  ( $\frac{\text{in-lb}}{\text{rad}}$ )

XS = \_\_\_\_\_, (0 assumed)

YS = \_\_\_\_\_, (0 assumed)

ZS = \_\_\_\_\_, (0 assumed)

TYS = \_\_\_\_\_, (0 assumed)

TZS = \_\_\_\_\_, (0 assumed)

TETRA

TYPE 5 PHYSICAL CONNECTING ELEMENT

UNCOUPLED POINT SPRING - DAMPER ELEMENT (CONTINUED)

2 1=damping  
 0=no damping  
 IDAMP=\_\_\_\_\_, (0 assumed)

If IDAMP=1, complete the input for one of the following two options:

Option 1: For damping based on Q-factor and selected frequency, enter the following (if QELEM is entered but not QFREQ, the frequency defined by the ISF variable on type A input sheet is used).

Q-factor  
 QELEM=\_\_\_\_\_,  
 frequency (hertz)  
 QFREQ=\_\_\_\_\_.

Option 2: For damping based on damping coefficient definition, enter the following:

$C_x \left( \frac{\text{lb-sec}}{\text{in}} \right)$	XD = _____, (0 assumed)
$C_y \left( \frac{\text{lb-sec}}{\text{in}} \right)$	YD = _____, (0 assumed)
$C_z \left( \frac{\text{lb-sec}}{\text{in}} \right)$	ZD = _____, (0 assumed)
$C_{\theta y} \left( \frac{\text{in-lb-sec}}{\text{rad}} \right)$	TYD= _____, (0 assumed)
$C_{\theta z} \left( \frac{\text{in-lb-sec}}{\text{rad}} \right)$	TZD= _____, (0 assumed)

#### 2.3.4.6 Squeeze Film Damper Element (Type H-3)

This input provides the capability to model a squeeze film damper. Unlike the other elements, which can be used for both transient and steady state analyses, the squeeze film damper element can only be used for transient analysis runs. If this input is present for a steady state analysis run, it is simply ignored. For further details about the squeeze film damper element, see reference 3.



TYPE 6 PHYSICAL CONNECTING ELEMENT  
(SQUEEZE FILM DAMPER ELEMENT)

Note: This input applies for transient analysis runs only. For steady state analysis runs, the below input is ignored.

2

\$  
\$LIST3  
ITYPE=6,  
ILEM=\_\_\_\_\_, element number

I end J end  
point number point number  
JT=\_\_\_\_\_, \_\_\_\_\_,

Note: The I end should correspond to the rotor or inner rotor. The J end should correspond to the stator or outer rotor.

BD=\_\_\_\_\_, nominal damper annulus diameter (in)

BL=\_\_\_\_\_, nominal damper annulus length (in)

BC=\_\_\_\_\_, damper annulus radial clearance (in)

VISC=\_\_\_\_\_, damper lubricant viscosity (reyns)

PVAP=\_\_\_\_\_, film rupture (vaporization) pressure (psi)

PB1=\_\_\_\_\_, specified boundary pressure at port 1 (psi)

PB2=\_\_\_\_\_, specified boundary pressure at port 2 (psi)

TH1=\_\_\_\_\_, angular position of port 1 (degrees)

TH2=\_\_\_\_\_, angular position of port 2 (degrees)

NGRID=\_\_\_\_\_, number of finite difference grid points per damper arc  
(should be odd)

NPORT=\_\_\_\_\_, number of lubricant ports (1 or 2)

NSOLN=\_\_\_\_\_, 1 = long bearing solution used  
2 = short bearing solution used (parabolic)

### 2.3.5. Restart, Time Integration, and Print/Plot Multiples (Type I-1)

First, the two lines \$ and \$LIST4 must be included regardless of whether the run is for a transient or a steady state analysis. The \$ is needed to signal to the program the end of the previous namelist section, and \$LIST4 signals the program the beginning of the LIST4 namelist section.

The next few input variables on the type I-1 input sheet apply for transient analysis runs only, and are ignored if the run is a steady state analysis run. If the run is a restart run, the user must enter ISTART=1 and, if desired, enter a restart time via the RTIME input variable. The restart time entered must correspond to one of the times for which output was printed on the initial run. If the run is a restart run and this variable is omitted, the program assumes that the restart time is the final time for which output was printed on the initial run (provided that the initial run did not terminate prematurely). Restart capability applies for transient analysis runs only. For more details about restart, see sections 2.1 and 3.1.

Next, for transient analysis runs the user must input the time step (DELTA input variable). The time step should be made equal to about 1/40 of the smallest period of oscillation. Making the time step much larger than this runs the risk of numerical problems.

Transient analysis runs begin at time equal 0 (unless the run is a restart run, in which case the run begins at the restart time), then time accumulates. The final time (TFINAL input variable) must be inputted so that the program knows when to stop. It is recommended that the user choose TFINAL such that the program will do a small number of time steps until the user is sure that his or her input is correct, so as to avoid possible costly garbage runs.

The next input variable is the transient analysis print multiple IPRMUL. This variable governs the number of time steps for which output is printed and for which data gets written onto the output restart file. If IPRMUL = 100, then one out of every 100 time steps computed is printed out and written onto the restart file. Similarly, the transient analysis plot multiple IPLMUL governs the number of time steps that get written onto the output plot file. If IPLMUL = 10, then one out of every 10 time steps is written onto the plot file.

Example: If DELTA=.0001 seconds, TFINAL=.5 seconds, IPRMUL=1000, and IPLMUL=20, then computations are made for 5001 time steps (starting with time = 0 and ending with time = .5). Output is printed for 6 time steps (0, .1, .2, .3, .4, and .5). Output is written to the plot file for 251 time steps (again starting at time = 0 and ending with time = .5).

It is recommended that the user pick IPRMUL such that no more than 10 time steps are printed, in order to avoid being buried in printed output. Actually, printout of about 3 time steps is normally sufficient. The printed output is useful to verify that there are no problems, but it is the plots that are most important. It is a good idea to check a couple of points on each plot produced versus the printed output to be sure the plots are correct.

The user should pick IPLMUL so that there are enough points (time steps) to define a smooth curve when the data is plotted. Usually about 200 or 300 points is enough to define a smooth curve. The user might need more points than this for an orbit type plot, especially if TFINAL is large. However, only very rarely would more than 1000 points be needed. The user is cautioned against requesting too many points (time steps) to be written to the plot file. The size of the output plot file is approximately proportional to the number of time steps included, and requesting too many points can make for very large plot files. Also, the more points on a plot file, the more time it takes for the plotting program to do its stuff.

The final variable on the type I-1 input sheet is the steady state analysis print multiple, JPRMUL. This variable only applies for steady state analysis runs, and is ignored for transient analysis runs. This variable governs the number of forcing frequencies for which output is printed for a steady state analysis run, and is similar to the transient analysis print multiple IPRMUL. If JPRMUL = 50, then output is printed for one out of every 50 of the forcing frequencies for which a solution is computed.

Note that for steady state analysis runs, all forcing frequencies are written to the plot file, so there is no steady state plot multiple analogous to the transient analysis plot multiple IPLMUL. This is because, for a transient analysis, one must find a solution for more time steps than are needed to produce a smooth curve in order to avoid numerical difficulties. However, for steady state runs there is no such restriction. For a steady state analysis, there is no reason for solving for any more forcing frequencies than are needed so that a smooth plot can later be obtained.

Note: The following two lines must be included regardless of whether the run is for a transient or a steady state analysis.

2

\$  
\$LIST4

### RESTART, TIME INTEGRATION, AND PRINT/PLOT MULTIPLES

Note: The below input applies for transient analysis runs only. For steady state analysis runs, the below input is ignored (a steady state run cannot be restarted).

0=new run  
1=restart run

ISTART=\_\_\_\_\_, restart indicator (integer) (0 assumed)

If the run is a restart run (ISTART=1), enter (if desired) the restart time. This time must correspond to one of the times for which output was printed on the initial run. If the run is a restart run and this input variable is omitted, the program assumes that the restart time is the final time for which output was printed on the initial run (provided that the initial run did not terminate prematurely).

RTIME=\_\_\_\_\_, restart time (seconds) for transient analysis runs

DELTA=\_\_\_\_\_, time step (seconds) for transient analysis runs

TFINAL=\_\_\_\_\_, final time (seconds) for transient analysis runs

IPRMUL=\_\_\_\_\_, print multiple for transient analysis runs (integer)

IPLMUL=\_\_\_\_\_, plot multiple for transient analysis runs (integer)

Note: The below input applies for steady state analysis runs only. For transient analysis runs, the below input is ignored. (For steady state analysis runs, all forcing frequencies are written to the plot file, so there is no steady state plot multiple).

JPRMUL=\_\_\_\_\_, print multiple for steady state analysis runs (integer)

\$ (Include if this is the last line of the input file, otherwise omit).

### 2.3.6 Steady State Nonlinear Analysis Input (Type I-2)

Input sheet I-2 applies only for a steady state analysis run in which at least one nonlinear type 3 physical connecting element (rub element) is present. For other runs, this input is ignored. Also, for each forcing frequency, the program makes a quick check to see if a rub is present (that is, the rub element dead band has been exceeded) for at least one of the rub elements. If no rub is present for any of the rub elements, then iteration is not needed to find the solution, and so this input is not used for that forcing frequency. See section 4.5.4 of volume 1 for further details.

First, the user may specify which subroutine from the IMSL library is to be used to solve the nonlinear equations (by iteration). If the user enters `IROUT = 1` or omits this variable, IMSL subroutine `ZSCNT` is used. If the user specifies `IROUT = 2` IMSL subroutine `ZSPOW` is used. `ZSCNT` is based on the secant method for solving nonlinear simultaneous equations. `ZSPOW` is based on Powell's hybrid algorithm, which is a variation of Newton's method. The authors' experience has been that both subroutines usually provide good results. Only for some cases with very large deflections (one inch or higher) were convergence problems encountered. For the few cases where convergence problems were encountered, subroutine `ZSCNT` usually worked better than `ZSPOW`.

Next, input variables `NSIG` and `ITMAX` control how many iterations are performed. Iteration ceases if two successive approximations to a given root agree in the first `NSIG` digits. Assumed values are 3 for `NSIG` and 200 for `ITMAX`. Neither input variable has much influence on program efficiency.

Finally, input variable `GUESS` can be used to input initial guesses for the rub element relative displacements (the unknowns in the iterating equations). If initial guesses are entered at all, they must be entered for all rub elements, and must be entered in the order that the rub elements appear in the input file. Of course, the initial guess input will only be used if the quick check that the program makes determines that iteration is needed, as outlined in section 4.5.4 of volume 1. The initial guess input also only applies for the very first solution (forced frequency). For a detailed discussion of how initial guesses are arrived at for all forced frequencies, see section 4.5.5 of volume 1.

If not entered via input variable `GUESS`, the initial guesses for the rub element relative displacements (if they are needed) will be those that would result assuming that there are no rub element forces for any of the rub elements. This is usually the best place to start anyway, so the `GUESS` input will usually not be needed. Sometimes, however, the user might decide to use the `GUESS` input to enter starting values that he or she has found out about from previous runs. More than one solution may be possible for a given forcing frequency, and with knowledge gained from previous runs, the user might use the `GUESS` input to steer the program to converge to the solution desired.

STEADY STATE NONLINEAR ANALYSIS INPUT

Note: The following input applies only for a steady state analysis in which at least one nonlinear type 3 physical connecting element (rub element) is present. Otherwise, the below input is ignored.

If desired, specify the subroutine to be used to solve the nonlinear equations.

IROUT=\_\_\_\_, 1 = IMSL subroutine ZSCNT (assumed value)  
2 = IMSL subroutine ZSPOW

Enter the iteration stopping criteria below. Iteration stops (a root is accepted) if either of the following criteria is satisfied.

NSIG=\_\_\_\_, A root is accepted if two successive approximations to a given root agree in the first NSIG digits (3 assumed)

ITMAX=\_\_\_\_, Maximum number of iterations (200 assumed)

Enter, if desired, initial guesses for the rub element relative displacements (the unknowns in the iterating equations). If initial guesses are entered at all, they must be entered for all rub elements. Enter one line (four values) for each rub element, in the same order that the rub elements appear in the input file. If not entered, the initial guesses will be those that would result assuming that there were no rub element forces for any of the rub elements. These initial guesses apply to the first solution (forced frequency) only.

RELATIVE DISPLACEMENT COMPONENT (INCHES)			
VERTICAL COS	HORIZONTAL COS	VERTICAL SIN	HORIZONTAL SIN

GUESS=

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

\$ (Include if this is the last line of the input file, otherwise omit).

### 2.3.7 Rotor Speed and Rate Input (Type J-1 and J-2)

Input sheets J-1 and J-2 are used for the rotor speed and rate input. This input is required if unbalance forces, gyroscopic forces, or nonstructural damping which is tied to the independent rotor speed is present. If this is not the case, the user can skip input sheets J-1 and J-2 altogether.

If rotor speed input is required, the user must specify which rotor is the independent rotor by setting IROTI to 1 or 2. Rotor 1 corresponds to subsystems 1, 2, and 3 and rotor 2 corresponds to subsystems 4, 5, and 6. The independent rotor is the rotor for which the BEGRPM, BEGTIM, and TRHIS input is entered for a transient analysis run, or for which the BRPM, FRPM, and SRPM input is entered for a steady state analysis run.

If rotor speed input is required and the run is a transient analysis run, the beginning time BEGTIM, beginning rpm BEGRPM, and time/rate table TRHIS must then be entered, which defines the speed and rate history for the independent rotor. There must be at least one and no more than 10 speed segments in the time/rate history table. However, if rotor speed input is required and the run is a steady state analysis run, the beginning rpm BRPM, final rpm FRPM, and step rpm SRPM must then be entered.

If rotor speed input is required and another rotor is also present, the other rotor is referred to as the dependent rotor, because its speed is a function of the independent rotor speed. For the dependent rotor, the user should enter the coefficients A, B, C, and D relating the dependent rotor speed Y (rpm) to that of the independent rotor speed X (rpm), where  $Y = AX^3 + BX^2 + CX + D$ .

ROTOR SPEED AND RATE INPUT

Note: This input is required if unbalance forces, gyroscopic forces, or nonstructural damping that is tied to the independent rotor speed is present.

Enter the independent rotor number (rotor for which the following input defining the speed applies). Permissible values are 0 (static case -unbalance forces, gyroscopic forces, and nonstructural damping which is tied to the independent rotor speed not considered), 1 (rotor corresponding to subsystems 1, 2, and 3), and 2 (rotor corresponding to subsystems 4, 5, and 6). Must be integer.

2

IROTI=\_\_\_\_, (0 assumed)

Note: The remainder of this page applies for transient analysis runs only. For steady state analyses, the below input is ignored.

If IROTI = 1 or IROTI = 2, enter the following:

BEGTIM=\_\_\_\_\_, beginning time for the first speed segment (applies to the independent rotor)

BEGRPM=\_\_\_\_\_, beginning speed (rpm) for the first speed segment (applies to the independent rotor)

If IROTI = 1 or IROTI = 2 enter the following table in chronological order (applies to the independent rotor ) (maximum of 10 segments):

Ending Time (seconds)	Rate (rpm/sec)
--------------------------	-------------------

TRHIS=

\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,  
\_\_\_\_\_, \_\_\_\_\_,

Segment 1
Segment 2
Segment 3
Segment 4
Segment 5
Segment 6
Segment 7
Segment 8
Segment 9
Segment 10

\$ (Include if this is the last line of the input file, otherwise omit).



ROTOR SPEED AND RATE INPUT (Continued)

Note: The following input applies for steady state analysis runs only. For transient analysis runs, the following input is ignored. Because (for steady state analysis runs only) all unbalance points must lie on the independent rotor, the exciting frequency will correspond to the independent rotor speed if unbalance loads are present.

If IROTI = 1 or IROTI = 2, enter the following:

2

BRPM=\_\_\_\_\_, beginning rpm for the independent rotor

FRPM=\_\_\_\_\_, final rpm for the independent rotor

SRPM=\_\_\_\_\_, step rpm

Note: The following input applies both for transient analysis and steady state analysis runs.

If IROTI = 1 or IROTI = 2 and a second rotor is present, input the following coefficients relating the second (dependent) rotor speed Y (rpm) to the independent rotor speed X (rpm), where  $Y = AX_3 + BX_2 + CX + D$ :

A=\_\_\_\_\_, (0 assumed)

B=\_\_\_\_\_, (0 assumed)

C=\_\_\_\_\_, (0 assumed)

D=\_\_\_\_\_, (0 assumed)

\$ (Include if this is the last line of the input file, otherwise omit).

### 2.3.8 Applied Loads (Type K-1 through M-2)

#### 2.3.8.1 Unbalance Loads (Type K-1 and K-2)

Input sheets K-1 and K-2 are for unbalance load input. To have unbalance loads, rotor speed input (type J-1 and J-2) is needed - otherwise, the unbalance load input is ignored.

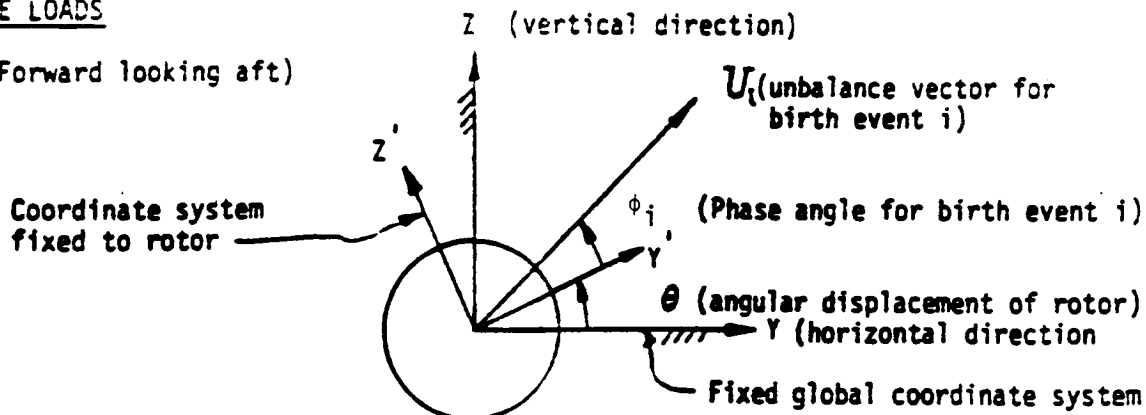
For transient analysis runs, the unbalance loads are entered via the UNBAL input variable. For each transient analysis unbalance load, four values must be entered - the time of birth (when the unbalance load becomes active), the global point number on the unbalanced rotor (the point must lie on rotor 1 or rotor 2), the magnitude of the unbalance (gm-in), and the phase angle (degrees) (see the figure on the type K-1 input sheet). For steady state analysis runs, on the other hand, the unbalance loads are entered via the UBALSS input variable. Only three values must be entered for each steady state unbalance load - the same values as for a transient analysis except that the time of birth is omitted. Whether the run is transient or steady state, there can be from 0 to a maximum of 20 unbalance loads.

The unbalance load input is quite flexible. For transient analyses, the time of birth can be the same for different birth events (time of birth does not apply for steady state analyses). Also, for both transient and steady state runs the same point can be referenced more than once if desired. If more than one unbalance load is specified for the same point, the unbalance loads are added together to get the total unbalance load. For example, the user could make a transient analysis run where a nominal unbalance load is applied at the engine fan location, then at a later time, after transient effects have died out, introduce a much larger unbalance due to a fan blade loss.

For a transient analysis restart run (steady state runs do not have restart capability), any unbalance loads that were present in the original run should be left in for the restart run. (Otherwise, the unbalance loads would disappear for the new run). Additional birth events that become active after the restart time may be added to the UNBAL input for a restart run, however.

APPLIED LOADS  
UNBALANCE LOADS

(Forward looking aft)



NOTE: To have unbalance loads, rotor speed input (type J-1 and J-2) is required - otherwise, the unbalance load input is ignored.

NOTE: The following input applies for transient analysis runs only. For steady state analysis runs, the following input is ignored.

If unbalance loads are desired, enter the following (maximum of 20 lines):

Time of birth (seconds)	Point number on unbalanced rotor	Magnitude (gm-in)	Phase Angle $\phi$ (degrees)
-------------------------	----------------------------------	-------------------	------------------------------

2

UNBAL =

_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____
_____	_____	_____	_____

If restart run and the time of birth is less than the restart time, then the unbalance load continues active for the restart run. Time of birth can be the same for different birth events. Also, the same point can be referenced more than once if desired.

\$ (Include if this is the last line of the input file, otherwise omit).

Note: The following applies for steady state analysis runs only. For transient analysis runs, the following input is ignored. Note that the variable name for steady state unbalance loads (UBALSS) is different than that for transient unbalance loads (UNBAL). Also note that all unbalance points must be located on the independent rotor (the independent rotor and its speed are specified on input sheets J-1 and J-2).

Global Point Number on Unbalanced (Independent) Rotor	Magnitude gm-in	Phase Angle $\phi$ degrees
---	--------------------	----------------------------------

2

[illegible]

64

#### 2.3.8.2 P\*cos( $\omega$ t) and P\*sin( $\omega$ t) Loads (Type L-1)

The P\*cos( $\omega$ t) and P\*sin( $\omega$ t) load input applies for transient analysis runs only. For steady state analysis runs, this input is ignored. If this type of input is not desired, this input sheet can be skipped altogether. If these loads are desired, the user must enter 5 values for each load via the CS input variable as shown on input sheet L-1. As for the unbalance load input, the same point can be referenced more than once, in which case the loads are added together to get the total load. There can be from 0 to a maximum of 30 loads.

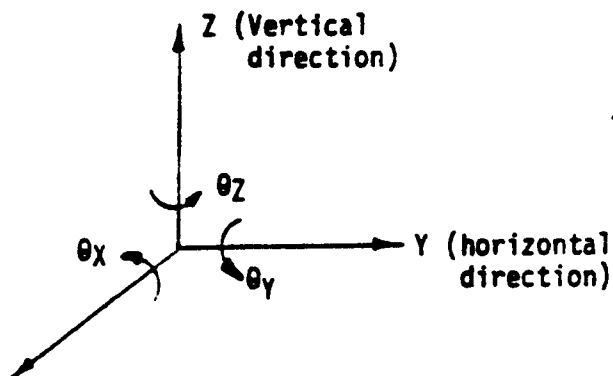
Definitions are as follows:

P = force amplitude, (lb or in-lb)

$\omega$  = frequency (hertz)

t = time (seconds)

$P = \cos(\omega t)$  and  $P = \sin(\omega t)$  LOADS



GLOBAL DIRECTION NUMBER	DIRECTION
1	Z
2	$\theta_Y$
3	Y
4	$\theta_Z$
5	X
6	$\theta_X$

X (axial direction, positive forward)

NOTE: The following applies for transient analysis runs only. For steady state analysis runs, the following input is ignored.

If  $P \cos(\omega t)$  or  $P \sin(\omega t)$  loads are desired, enter the following (maximum of 30 lines):

GLOBAL POINT NUMBER	TYPE (1=COS 2=SIN)	AMPLITUDE P (1b or in-1b)	FREQUENCY $\omega$ (hertz)	GLOBAL DIRECTION NUMBER
---------------------------	--------------------------	---------------------------------	----------------------------------	-------------------------------

CS =

[illegible]

\$ (Include if this is the last line of the input file, otherwise omit).

#### 2.3.8.3 $P \cos(\omega t + \phi)$ Loads (Type L-2)

$P \cos(\omega t + \phi)$  loads apply for steady state analysis runs only. For transient analysis runs, this input is ignored. If this type of loading is not desired, the type L-2 input sheet can be skipped altogether. If this type of loading is desired, 4 values are entered for each load via the COSSS input variable, as shown on input sheet L-2. The same point may be referenced more than once, in which case the loads are added together to get the total load. There can be from 0 to a maximum of 30 loads.

Definitions are as follows:

$P$  = force amplitude (lb or in-lb)

$\omega$  = forcing frequency (hertz)

$t$  = time (seconds)

$\phi$  = phase angle (degrees)

APPLIED LOADSP\*cos( $\omega t + \phi$ ) LOADS

Note: The following input applies for steady state analysis runs only. For transient analysis runs, the below input is ignored.

If P\*cos( $\omega t + \phi$ ) loads are desired, input the following (maximum of 30 lines):

Global Point Number	Amplitude P lb or in-lb	Phase Angle $\phi$ degrees	Global Direction Number
---------------------------	-------------------------------	----------------------------------	-------------------------------

2

COSSS=

_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____
_____	,	_____	,	_____	,	_____

If steady state analysis run and unbalance loads (input sheet K-2) are present in addition to P\*cos( $\omega t + \phi$ ) loads, the program will set the forcing frequency for the P\*cos( $\omega t + \phi$ ) loads equal to the speed of the unbalance (independent) rotor, and the below input, if present, is overridden. If, however, P\*cos( $\omega t + \phi$ ) loads are present but not unbalance loads, you must enter the below input to define the forcing frequency  $\omega$  for the P\*cos( $\omega t + \phi$ ) loads. A steady state analysis will then be performed at each of the below frequencies for every rotor speed defined on the type J-2 input sheet.

BW=\_\_\_\_\_, beginning forcing frequency (rpm)

FW=\_\_\_\_\_, final forcing frequency (rpm)

SW=\_\_\_\_\_, step forcing frequency (rpm)

\$ (Include if this is the last line of the input file, otherwise omit).

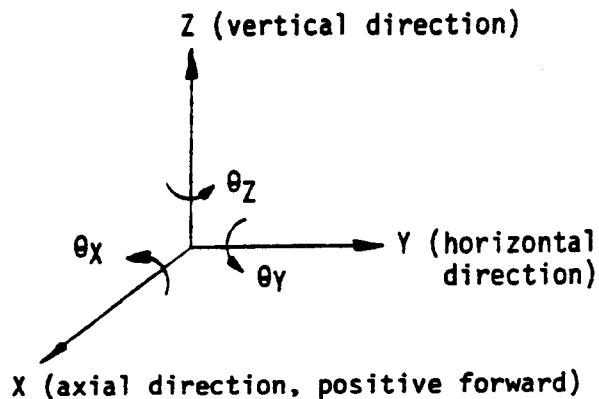


#### 2.3.8.4 Time-Force History Loads (Type M-1 and M-2)

Input sheet M-1 and M-2 apply for transient analysis runs only. For steady state analysis runs, this input is ignored. If time-force history loads are not desired, input sheets M-1 and M-2 can be skipped altogether.

If time-force history loads are desired, the user must specify the global point number, global direction number, and table number for each time-force history load via the NTF input variable on input sheet M-1 (values must be integers). The table number must be a value between 1 and 10. There can be from 0 to a maximum of 30 time-force history loads entered in the NTF array.

Then, for each table number referenced in the NTF array, the array table is entered via TABLE(1,1,NT), where NT is the table number referenced in the NTF array. Entries in each table must be in order of increasing time. If the first time-force entry in the table is for a time other than zero, then the force is assumed to be zero up to the time of the first table entry. If the last time-force pair in the table is for a time less than the final time of the run, then for times greater than that of the last table entry the force is assumed constant and equal to the force for the last table entry. A table may have only one time-force pair entry, in which case the force is zero till the entered time and afterward equal to the entered force. There can be a maximum of 10 tables, and a maximum of 10 time-force pairs in each table.

APPLIED LOADS  
TIME-FORCE HISTORY LOADS

GLOBAL DIRECTION NUMBER	DIRECTION
1	Z
2	$\theta_y$
3	Y
4	$\theta_z$
5	X
6	$\theta_x$

Note: Sheets M-1 and M-2 are for transient analysis runs only. For steady state analysis runs, this input is ignored.

If time-force history loads are desired, enter the following (values must be integers) (maximum of 30 lines):

GLOBAL POINT NUMBER	GLOBAL DIRECTION NUMBER	TABLE NUMBER NT (VALUE BETWEEN 1 & 10)
---------------------------	-------------------------------	--

2

NTF =

\_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,  
 \_\_\_\_\_, \_\_\_\_\_, \_\_\_\_\_,

APPLIED LOADS  
TIME-FORCE HISTORY LOADS (Continued)

For each time-force table, enter the following. Entries must be in order of increasing time. Substitute the actual table number in place of the dimension NT below. A maximum of 10 tables and 10 time-force pairs in each table is permitted.

TIME	FORCE
seconds	lb or in-lb

2

TABLE(1,1,NT)=

_____	_____
_____	_____
_____	_____
_____	_____
_____	_____
_____	_____
_____	_____
_____	_____
_____	_____
_____	_____
_____	_____

\$ (Include if this is the last line of the input file, otherwise omit).

### 2.3.9 Gyroscopic Loads (Type N)

Input sheet N is for the gyroscopic load input. This input applies for both transient and steady state analysis runs. For each gyroscopic load, the point number on the rotor and the polar moment of inertia ( $\text{lb-in}^2$ ) is entered. There can be a maximum of 30 gyroscopic loads. Since this input models the cross-axis coupling forces associated with Coriolis acceleration, both the vertical and horizontal subsystems must be included for the rotor(s).

GYROSCOPIC LOADS

Note: To have gyroscopic loading, rotor speed input (type J-1 and J-2 input sheets) must be included. Otherwise, the gyroscopic load input is ignored.

Note: For gyroscopic loading, both the vertical and horizontal subsystems must be included for the rotor(s).

If gyroscopic loads are desired, enter the following (maximum of 30 lines):

POINT NUMBER ON ROTOR	POLAR MOMENT OF INERTIA $I_p$ lb-in <sup>2</sup>
-----------------------------	--

2

GYRO=

_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,
_____	,	_____	,

\$ (Include is this is the last line of the input file, otherwise omit).

### 2.3.10 Data for Plot file (Type P-1 and P-2)

Input sheets P-1 and P-2 are used to control what data (if any) gets written to the output plot file. These input sheets apply for both transient and steady state analysis runs.

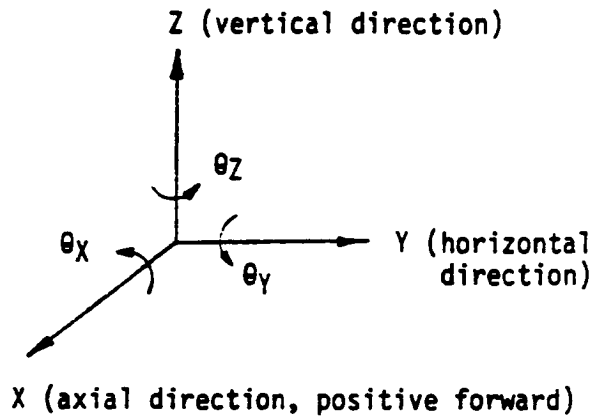
First comes input variable Iplot. If no output plot file is wanted, the user should set variable Iplot to 0 and dispense with the rest of input sheets P-1 and P-2. If, however, the user sets Iplot to 1 or omits this input variable, then an output plot file will be produced, and the user should complete the rest of input sheets P-1 and P-2.

The plot file size in many cases would be huge if all the data that conceivably might be desired were written onto the plot file. For this reason, the NPD (input sheet P-1) and NEPD (input sheet P-2) input variables were added. This input allows the user to specify the data that he or she wants written to the plot file. Be sure to include this input for all the data that you might want to plot.

The NPD array allows the user to select the point and direction pairs for which the coordinates, displacement magnitudes, velocity magnitudes, and modal force magnitudes get written to the output plot file. For steady state analyses, the displacement phase angles, velocity phase angles, and modal force phase angles for these point and direction pairs are also written to the output plot file. If the user doesn't want to plot values for any point and direction pairs, then the NPD array should be omitted. A maximum of 50 point and direction pairs may be specified.

The NEPD array allows the user to select the physical connecting element number, global point number, and global direction number trio for which the physical connecting element force magnitudes are written to the output plot file. For steady state analyses, the physical connecting element force phase angle for these trios are also written to the output plot file. Again, if the user doesn't want to plot values for any element, point, and direction trios, he or she should omit this array. A maximum of 50 element, point, and direction trios may be specified.

In addition, certain other data is always automatically written to the output plot file (provided that an output plot file is generated). This includes the time (transient analysis only), forcing frequency (steady state analysis only), independent and dependent rotor speed, independent and dependent rotor angular displacement (transient analysis only), generalized displacement magnitudes for the flexible bladed disk modes, phase angles for the flexible bladed disk modes (steady state analysis only), data for the flexible bladed disk(s) (if any were present in the TETRA run), and data for the rub elements (if any were present in the TETRA run). For further details about the contents of the output plot file, see section 3.2.

DATA FOR PLOT FILE

GLOBAL DIRECTION NUMBER	DIRECTION
1	Z
2	$\theta_y$
3	Y
4	$\theta_z$
5	X
6	$\theta_x$

2

I PLOT=\_\_\_\_, 0 = no plot file  
1 = output plot file produced (assumed value)

Enter the following (point,direction) pairs for which data is to be written to the output plot file. For both transient and steady state analysis runs this data includes the coordinates, displacement magnitudes, velocity magnitudes, and modal force magnitudes. For steady state analysis runs, this data also includes the displacement phase angle, velocity phase angle, and modal force phase angle. Values must be integers. Maximum of 50 lines.

GLOBAL POINT NUMBER	GLOBAL DIRECTION NUMBER
---------------------------	-------------------------------

NPD=

\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,  
\_\_\_\_,\_\_\_\_,

\$ (Include is this is the last line of the input file, otherwise omit).

DATA FOR PLOT FILE (Continued)

Enter the following (element,point,direction) trios (if any) for which data is to be written to the output plot file. For both transient and steady state analysis runs, the physical connecting element force magnitude is written to the output plot file for these trios. For steady state analysis runs, the physical connecting element force phase angle is also written to the output plot file for these trios. Values must be integers. Maximum of 50 lines.

PHYSICAL CONNECTING ELEMENT NUMBER	GLOBAL POINT NUMBER	GLOBAL DIRECTION NUMBER
---	---------------------------	-------------------------------

2

NEPD=

_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____
_____	_____	_____

\$ (Include if this is the last line of the input file, otherwise omit).



### 3.0 TETRA 2 OUTPUT

#### 3.1 Output Restart File

If the run is for a transient analysis, an output restart file will be generated. This file is written to file code 24, so for transient analysis runs the user must have a line or two in his or her JCL (job control language) that assigns a name to the output restart file and associates it with file code 24. The output restart file thus produced may then subsequently be used as the input restart file for a new transient analysis restart run.

For more information on the restart file, see section 2.1.

### 3.2 OUTPUT PLOT FILE

Usually the user requests a plot file to be written, so that a postprocessing program can later be run to plot the results (plots are vital for interpreting the results from this program). The input variable IPLOT on input sheet P-1 controls whether or not a plot file is produced (this default is to produce one).

The plot file is an unformatted sequential file. This plot file is written to file code 23. Therefore, if a plot file has been requested, the user must add a line or two to his or her JCL (job control language) to assign a name to the plot file and to associate the plot file with file code 23.

The plot file has the potential of being a very large file, depending on how many time steps (for a transient analysis run) or how many forcing frequencies (for a steady state analysis run) are written to it. One means by which the size of the plot file is limited is that the user specifies (via input sheets P-1 and P-2) what data is to be written to this file. For more details about this, see the discussion of the type P-1 and P-2 input sheets.

For a transient analysis run, the user can also limit the size of the plot file by controlling how many of the time steps analyzed are written to the plot file via the input plot multiple IPLMUL on the type I-1 input sheet. See the discussion of the type I-1 input sheet for advice on what value to use for IPLMUL. For a steady state analysis run, on the other hand, all of the forcing frequencies analyzed are written to the plot file (provided, of course, that the user requested a plot file). The reason that a plot multiple is provided for a transient analysis only is that for a transient analysis the user must run more time steps than needed for plotting to avoid numerical problems, whereas there is no such requirement for a steady state analysis. In other words, for a steady state analysis, there is no good reason for analyzing any more forcing frequencies than needed for the plot file so that a smooth plot can be later obtained.

The plot file contents may be divided into sections, of which there are three types as follows. The first type of section is the plot file preface (see table 3-1), which comes first and contains information telling the plot program what type of run (transient or steady state) and what information is present on the plot file (which points, directions, forces, etc.). This section is present for both transient and steady state analyses. This section occurs only once, and every line is a record for this section.

The second section is for the transient analysis plot data (see table 3-2), and is present only if the TETRA run that generated the plot file was a transient analysis run. This section is repeated for each time step written to the plot file. The whole section (that is, each time step) is a record.

Finally, the third section is for the steady state analysis plot data (see table 3-3), and is present only if the TETRA run that generated the plot file was a steady state analysis run. This section is repeated for each forcing frequency

that is written to the plot file. The whole section (that is, each forcing frequency) is a record.

Definitions of the plot file variables contained in all three sections (arranged alphabetically) are in table 3-4.

Note that the first two types of sections (the plot file preface and the transient analysis plot data) are almost identical to what they were in the original TETRA program (which could only perform transient analyses). The only difference to the plot file preface is the addition of the variables NTYPE, IFBDR, and TINT which are tacked onto the end of the first line of the plot file preface. The only difference for the transient analysis plot file section is the addition of the variables DVM and DHM (which are useful in making orbit type plots). Thus, plot programs which were written for the original TETRA program (which was for transient analyses only) will have to be modified only slightly to work for transient analysis TETRA 2 runs, but will have to be modified much more extensively if they are also to work for steady state analysis TETRA 2 runs.

Table 3-1. Plot file preface

NP,NL1,NL2,NRE,NEL,NPLOTP,NTYPE,IFBDR,TINT  
 IPLOT<sub>1</sub>,IDPLOT<sub>1</sub>,XPT<sub>1</sub>,YPT<sub>1</sub>,ZPT<sub>1</sub>

.  
 .  
 .  
 .

IPLOT<sub>NP</sub>,IDPLOT<sub>NP</sub>,XPT<sub>NP</sub>,YPT<sub>NP</sub>,ZPT<sub>NP</sub>  
 LFBD1<sub>1</sub>,RFBD1<sub>1</sub>,AFBD1<sub>1</sub>

.  
 .  
 .  
 .

LFBD1<sub>NL1</sub>,RFBD1<sub>NL1</sub>,AFBD1<sub>NL1</sub>  
 LFBD2<sub>1</sub>,RFBD2<sub>1</sub>,AFBD2<sub>1</sub>

.  
 .  
 .  
 .

LFBD2<sub>NL2</sub>,RFBD2<sub>NL2</sub>,AFBD2<sub>NL2</sub>  
 ILEM3<sub>1</sub>,ILEM3<sub>1</sub>,ILEM3<sub>1</sub>

.  
 .  
 .  
 .

ILEM3<sub>NRE</sub>,ILEM3<sub>NRE</sub>,ILEM3<sub>NRE</sub>  
 ILEM<sub>1</sub>,IPT<sub>1</sub>,IDIR<sub>1</sub>

.  
 .  
 .  
 .

ILEM<sub>NEL</sub>,IPT<sub>NEL</sub>,IDIR<sub>NEL</sub>

Table 3-2. Plot file data for a transient analysis run

```

TIME,RPMI,RPMD,THETA1,THETAD
PMAG1,QMAG1,PMAG2,QMAG2
XMAG1,VMAG1,FMODM1
.
.
.
XMAGNP,VMAGNP,FMODMNP
PSI11,UMAG11,VMAG11,S1FBD11,S2FBD11,S3FBD11
.
.
.
PSI1NL1,UMAG1NL1,VMAG1NL1,S1FBD1NL1,S2FBD1NL1,S3FBD1NL1
PSI21,UMAG21,VMAG21,S1FBD21,S2FBD21,S3FBD21
.
.
.
PSI2NL2,UMAG2NL2,VMAG2NL2,S1FBD2NL2,S2FBD2NL2,S3FBD2NL2
RD1,CL1,F1,DVM1,DHM1
.
.
.
RDNRE,CLNRE,FNRE,DVMNRE,DHMNRE
FMAG1
.
.
.
FMAGNEL

```

Table 3-3. Plot file data for a steady state analysis run

```

WFF,RPMI,RPMD
PMAG1,PPH1,QMAG1,QPH1
PMAG2,PPH2,QMAG2,QPH2
XMAG1,XPHASE1,VMAG1,VPHASE1,FMODM1,FMODPH1
.
.
.
.
XMAGNp,XPHASENp,VMAGNp,VPHASENp,FMODMNp,FMODPHNp
UMAGP11,VMAGP11,SIG1P11,SIG2P11,SIG3P11
.
.
.
.
UMAGP1NL1,VMAGP1NL1,SIG1P1NL1,SIG2P1NL1,SIG3P1NL1
UMAGQ11,VMAGQ11,SIG1Q11,SIG2Q11,SIG3Q11
.
.
.
.
UMAGQ1NL1,VMAGQ1NL1,SIG1Q1NL1,SIG2Q1NL1,SIG3Q1NL1
UMAGP21,VMAGP21,SIG1P21,SIG2P21,SIG3P21
.
.
.
.
UMAGP2NL2,VMAGP2NL2,SIG1P2NL2,SIG2P2NL2,SIG3P2NL2
UMAGQ21,VMAGQ21,SIG1Q21,SIG2Q21,SIG3Q21
.
.
.
.
UMAGQ2NL2,VMAGQ2NL2,SIG1Q2NL2,SIG2Q2NL2,SIG3Q2NL2
RX1,RN1,CX1,CN1,FX1,FN1,DVM1,DVP1,DHM1,DHP1
.
.
.
.
RXNRE,RNNRE,CXNRE,CNNRE,FXNRE,FNNRE,DVMNRE,DVPNRE,DHMNRE,DHPNRE
FMAG1FPHASE1
.
.
.
.
FMAGNEL,FPHASENEL

```

Table 3-4 Plot file variable definitions (alphabetical order)

<u>VARIABLE</u>	<u>DEFINITION</u>
AFBD1 <sub>i</sub>	Axial coordinate for the i'th local point on flexible bladed disk number 1. Applies for both transient and steady state analyses.
AFBD2 <sub>i</sub>	Axial coordinate for the i'th local point on flexible bladed disk number 2. Applies for both transient and steady state analyses.
CL <sub>i</sub>	Clearance (inches) for the i'th type 3 physical connecting element (rub element). Applies for a transient analysis run only.
CN <sub>i</sub>	Minimum clearance (inches) for the i'th type 3 physical connecting element (rub element). Applies for a steady state analysis run only.
CX <sub>i</sub>	Maximum clearance (inches) for the i'th type 3 physical connecting element (rub element). Applies for a steady state analysis run only.
DHM <sub>i</sub>	Magnitude (inches) of the difference between the horizontal direction displacement at joint I minus that of joint J for the i'th rub element. Applies for both transient and steady state analysis runs.
DHP <sub>i</sub>	Phase angle (degrees) for the difference between the horizontal direction displacement at joint I minus that of joint J for the i'th rub element. Applies for steady state analysis runs only.
DVM <sub>i</sub>	Magnitude (inches) of the difference between the vertical direction displacement at joint I minus that of joint J for the i'th rub element. Applies for both transient and steady state analysis runs.
DVP <sub>i</sub>	Phase angle (degrees) for the difference between the vertical direction displacement at joint I minus that of joint J for the i'th rub element. Applies for steady state analysis runs only.
F <sub>i</sub>	Force magnitude (pounds) for the i-th type 3 physical connecting element (rub element). Applies for a transient analysis run only.
FMA <sub>i</sub>	Force magnitude (pounds) for the i'th (element,point,direction) trio. Applies for both transient and steady state analysis runs.
FMODM <sub>i</sub>	Modal force magnitude (lb or in-lb) for the i'th (point,direction) pair. Applies for both transient and steady state analyses.
FMODPH <sub>i</sub>	Modal force phase angle (degrees) for the i'th (point,direction) pair. Applies for steady state analysis runs only.
FN <sub>i</sub>	Minimum harmonically averaged force magnitude (pounds) for the i'th type 3 physical connecting element. Applies for a steady state analysis run only.
FPHASE <sub>i</sub>	Phase angle (degrees) for the i'th (element,point,direction) trio. Applies for steady state analysis runs only.
FX <sub>i</sub>	Maximum harmonically averaged force magnitude (pounds) for the i'th type 3 physical connecting element. Applies for a steady state analysis run only.
IDIR <sub>i</sub>	Global direction number for the i'th (element,point,direction) trio for which data is written to the plot file. Applies for both transient and steady state analysis runs.
IDPLOT <sub>i</sub>	Global direction number for the i'th (point,direction) pair for which data is written to the plot file. Applies for both transient and steady state analyses.

IFBDR Indicator telling which rotor contains the flexible bladed disks. Permissible values are 0 (no flexible bladed disks), 1 (flexible bladed disks are on the independent rotor), and 2 (flexible bladed disks are on the dependent rotor).

ILEM<sub>i</sub> Element number for the i'th (element,point,direction) trio for which data is written to the plot file. Applies for both transient and steady state analyses.

ILEM3<sub>i</sub> Element number for the i'th type 3 physical connecting element (rub element). Applies for both transient and steady state analyses.

IPLOT<sub>i</sub> Global point number for the i'th (point,direction) pair for which data is written to the plot file. Applies for both transient and steady state analyses.

IPT<sub>i</sub> Global point number for the i'th (element,point,direction) trio for which data is written to the plot file. Applies for both transient and steady state analysis runs.

LFBD1<sub>i</sub> Local point number for the i'th local point on flexible bladed disk number 1. Applies for both transient and steady state analyses.

LFBD2<sub>i</sub> Local point number for the i'th local point on flexible bladed disk number 2. Applies for both transient and steady state analyses.

NEL Number of (element,point,direction) trios for which data is written to the plot file. Applicable for both transient and steady state analysis runs.

NL1 Number of local points on flexible bladed disk number 1. Applicable for both transient and steady state analysis runs.

NL2 Number of local points on flexible bladed disk number 2. Applicable for both transient and steady state analysis runs.

NP Number of (point,direction) pairs for which data is written to the plot file. Applicable for both transient and steady state analysis runs.

NPLOT<sub>P</sub> Number of plot points that should have been written to the plot file. For a transient analysis, this equals the number of time steps that should have been written to the plot file. For a steady state analysis, this equals the number of forcing frequencies that should have been written to the plot file. If for any reason the TETRA run terminated prematurely, the actual number of plot points might be less than this number.

NRE Number of type 3 physical connecting elements (rub elements). Applicable for both transient and steady state analysis runs.

NTYPE Analysis type code, where NTYPE=0 signifies a transient analysis run, and NTYPE=1 signifies a steady state analysis run.

PMAG1 Generalized displacement magnitude for the horizontal nodal diameter mode for flexible bladed disk number 1. Applies for both transient and steady state analyses.

PMAG2 Generalized displacement magnitude for the horizontal nodal diameter mode for flexible bladed disk number 2. Applies for both transient and steady state analyses.

PPH1 Phase angle (degrees) for the generalized displacement for the horizontal nodal diameter mode for flexible bladed disk number 1. Applies for a steady state analysis run only.

PPH2 Phase angle (degrees) for the generalized displacement for the



horizontal nodal diameter mode for flexible bladed disk number 2. Applies for a steady state analysis run only.

PSI1<sub>i</sub> Angle psi (degrees) for the i'th local point on flexible bladed disk number 1. Applies for a transient analysis run only.

PSI2<sub>i</sub> Angle psi (degrees) for the i'th local point on flexible bladed disk number 2. Applies for a transient analysis run only.

QMAG1 Generalized displacement magnitude for the vertical nodal diameter mode for flexible bladed disk number 1. Applies for both transient and steady state analyses.

QMAG2 Generalized displacement magnitude for the vertical nodal diameter mode for flexible bladed disk number 2. Applies for both transient and steady state analyses.

QPH1 Phase angle (degrees) for the generalized displacement for the vertical nodal diameter mode for flexible bladed disk number 1. Applies for a steady state analysis run only.

QPH2 Phase angle (degrees) for the generalized displacement for the vertical nodal diameter mode for flexible bladed disk number 2. Applies for a steady state analysis run only.

RD<sub>i</sub> Relative displacement magnitude for the i'th type 3 physical connecting element (rub element). Applies for a transient analysis run only.

RFBD1<sub>i</sub> Radial coordinate for the i'th local point on flexible bladed disk number 1. Applies for both transient and steady state analysis runs.

RFBD2<sub>i</sub> Radial coordinate for the i'th local point on flexible bladed disk number 2. Applies for both transient and steady state analysis runs.

RN<sub>i</sub> Minimum relative displacement magnitude for the i'th type 3 physical connecting element (rub element). Applies for a steady state analysis run only.

RPMD Speed (rpm) of the dependent rotor. Applicable for both transient and steady state analyses.

RPMI Speed (rpm) of the independent rotor. Applicable for both transient and steady state analyses.

RX<sub>i</sub> Maximum relative displacement magnitude for the i'th type 3 physical connecting element (rub element). Applies for a steady state analysis run only.

S1FBD1<sub>i</sub> SIGMA1 stress (psi) for the i'th local point on flexible bladed disk number 1. Applies for a transient analysis run only.

S1FBD2<sub>i</sub> SIGMA1 stress (psi) for the i'th local point on flexible bladed disk number 2. Applies for a transient analysis run only.

S2FBD1<sub>i</sub> SIGMA2 stress (psi) for the i'th local point on flexible bladed disk number 1. Applies for a transient analysis run only.

S2FBD2<sub>i</sub> SIGMA2 stress (psi) for the i'th local point on flexible bladed disk number 2. Applies for a transient analysis run only.

S3FBD1<sub>i</sub> SIGMA3 stress (psi) for the i'th local point on flexible bladed disk number 1. Applies for a transient analysis run only.

S3FBD2<sub>i</sub> SIGMA3 stress (psi) for the i'th local point on flexible bladed disk number 2. Applies for a transient analysis run only.

SIG1P1<sub>i</sub> SIGMA1 stress (psi) for the horizontal nodal diameter mode for the i'th local point on flexible bladed disk number 1. Applies for a steady state analysis run only.

SIG1P2 <sub>i</sub>	SIGMA1 stress (psi) for the horizontal nodal diameter mode for the i'th local point on flexible bladed disk number 2. Applies for a steady state analysis run only.
SIG1Q1 <sub>i</sub>	SIGMA1 stress (psi) for the vertical nodal diameter mode for the i'th local point on flexible bladed disk number 1. Applies for a steady state analysis run only.
SIG1Q2 <sub>i</sub>	SIGMA1 stress (psi) for the vertical nodal diameter mode for the i'th local point on flexible bladed disk number 2. Applies for a steady state analysis run only.
SIG2P1 <sub>i</sub>	SIGMA2 stress (psi) for the horizontal nodal diameter mode for the i'th local point on flexible bladed disk number 1. Applies for a steady state analysis run only.
SIG2P2 <sub>i</sub>	SIGMA2 stress (psi) for the horizontal nodal diameter mode for the i'th local point on flexible bladed disk number 2. Applies for a steady state analysis run only.
SIG2Q1 <sub>i</sub>	SIGMA2 stress (psi) for the vertical nodal diameter mode for the i'th local point on flexible bladed disk number 1. Applies for a steady state analysis run only.
SIG2Q2 <sub>i</sub>	SIGMA2 stress (psi) for the vertical nodal diameter mode for the i'th local point on flexible bladed disk number 2. Applies for a steady state analysis run only.
SIG3P1 <sub>i</sub>	SIGMA3 stress (psi) for the horizontal nodal diameter mode for the i'th local point on flexible bladed disk number 1. Applies for a steady state analysis run only.
SIG3P2 <sub>i</sub>	SIGMA3 stress (psi) for the horizontal nodal diameter mode for the i'th local point on flexible bladed disk number 2. Applies for a steady state analysis run only.
SIG3Q1 <sub>i</sub>	SIGMA3 stress (psi) for the vertical nodal diameter mode for the i'th local point on flexible bladed disk number 1. Applies for a steady state analysis run only.
SIG3Q2 <sub>i</sub>	SIGMA3 stress (psi) for the vertical nodal diameter mode for the i'th local point on flexible bladed disk number 2. Applies for a steady state analysis run only.
THETAD	Angular displacement (revolutions) for the dependent rotor. Applicable for transient analyses only.
THETAI	Angular displacement (revolutions) for the independent rotor. Applicable for transient analyses only.
TIME	Time (seconds). Applicable for a transient analysis only.
TINT	Time interval (seconds) between the time steps on the plot file. This variable applies for a transient analysis run only. For a steady state analysis, this variable is present on the plot file but equals 0.
UMAG1 <sub>i</sub>	Displacement u (inches) for the i'th local point on flexible bladed disk number 1. Applies for transient analysis runs only.
UMAG2 <sub>i</sub>	Displacement u (inches) for the i'th local point on flexible bladed disk number 2. Applies for transient analysis runs only.
UMAGP1 <sub>i</sub>	Displacement magnitude u (inches) for the horizontal nodal diameter mode for the i'th point on flexible bladed disk number 1. Applies for steady state analysis runs only.
UMAGP2 <sub>i</sub>	Displacement magnitude u (inches) for the horizontal nodal diameter

mode for the i'th point on flexible bladed disk number 2. Applies for steady state analysis runs only.

UMAGQ1<sub>i</sub> Displacement magnitude u (inches) for the vertical nodal diameter mode for the i'th point on flexible bladed disk number 1. Applies for steady state analysis runs only.

UMAGQ2<sub>i</sub> Displacement magnitude u (inches) for the vertical nodal diameter mode for the i'th point on flexible bladed disk number 2. Applies for steady state analysis runs only.

VMAG<sub>i</sub> Velocity magnitude (in/sec or rad/sec) for the i'th (point,direction) pair. Applies for both transient and steady state analyses.

VMAG1<sub>i</sub> Displacement v (inches) for the i'th local point on flexible bladed disk number 1. Applies for transient analysis runs only.

VMAG2<sub>i</sub> Displacement v (inches) for the i'th local point on flexible bladed disk number 2. Applies for transient analysis runs only.

VMAGP1<sub>i</sub> Displacement magnitude v (inches) for the horizontal nodal diameter mode for the i'th point on flexible bladed disk number 1. Applies for steady state analysis runs only.

VMAGP2<sub>i</sub> Displacement magnitude v (inches) for the horizontal nodal diameter mode for the i'th point on flexible bladed disk number 2. Applies for steady state analysis runs only.

VMAGQ1<sub>i</sub> Displacement magnitude v (inches) for the vertical nodal diameter mode for the i'th point on flexible bladed disk number 1. Applies for steady state analysis runs only.

VMAGQ2<sub>i</sub> Displacement magnitude v (inches) for the vertical nodal diameter mode for the i'th point on flexible bladed disk number 2. Applies for steady state analysis runs only.

VPHASE<sub>i</sub> Velocity phase angle (degrees) for the i'th (point,direction) pair. Applies for a steady state analysis only.

WFF Steady state forcing frequency (rpm). Applicable for a steady state analysis only.

XMAG<sub>i</sub> Displacement magnitude (inches or radians) for the i'th (point,direction) pair. Applies for both transient and steady state analyses.

XPHASE<sub>i</sub> Displacement phase angle (degrees) for the i'th (point,direction) pair. Applies for steady state analyses only.

XPT<sub>i</sub> X coordinate for the i'th (point,direction) pair for which data is written to the plot file. Applies for both transient and steady state analyses.

YPT<sub>i</sub> Y coordinate for the i'th (point,direction) pair for which data is written to the plot file. Applies for both transient and steady state analyses.

ZPT<sub>i</sub> Z coordinate for the i'th (point,direction) pair for which data is written to the plot file. Applies for both transient and steady state analyses.

### 3.3 Printed Output

For every TETRA run, printed output is produced. The file code for the printed output is the standard 06 file.

The size of the printed output depends on the value inputted for the printout option variable IOUT on type A input sheet. See the discussion of the type A input for advice on what value to use for IOUT. For a transient analysis, the size of the printed output also depends greatly on the value inputted for the transient analysis print multiple IPRMUL on type I-1 input sheet. Likewise, for a steady state analysis, the size of the printed output depends greatly on the value inputted for the steady state analysis print multiple JPRMUL, also on type I-1 input sheet. See the discussion of the type I-1 input sheet for advice on what values to use for IPRMUL and JPRMUL.

For a sample printed output listing, see section 5.3.1. This section contains a listing of the printed output for the demonstrator model with the standard form of the printout (IOUT=1 on type A input sheet) requested.

#### 4.0 PROGRAM STRUCTURE AND SUBROUTINES

TETRA 2 represents a substantial growth in program size as well as program capability. The previous version of TETRA (including flexible bladed disk capability but with only transient analysis capability and not steady state analysis capability) consisted of a main routine plus 36 subroutines. TETRA 2 consists of a main routine plus 87 subroutines. Not included in this total are 5 IMSL library subroutines which are called directly from TETRA subroutines, and several additional IMSL library subroutines which are not called directly from TETRA subroutines, but rather from the other IMSL library subroutines in TETRA. In terms of lines of coding, the previous version of TETRA had about 8000 lines, while TETRA 2 has about 30000 lines.

The large increase in program size is largely because the steady state analysis portion of TETRA proved to be more complex and requiring more steps than the transient analysis portion of TETRA. In particular, a large amount of coding was needed for the assembly of the large global matrices, which are required for steady state analyses but not for transient analyses. The coding for the nonlinear rub element capability also proved to be much more extensive for the steady state analysis portion than for the transient analysis portion, with a lot of coding needed to assemble the iterating equations and perform harmonic averaging. Also contributing to the increase in program size was additional capability added to TETRA 2, such as expanding the rub element to include a nonlinear cubic factor, addition of structural damping capability, added printout options, etc. Contributing to the large increase in the number of lines is the fact that more extensive documentation for each subroutine is included in the coding (primarily at the beginning of each subroutine).

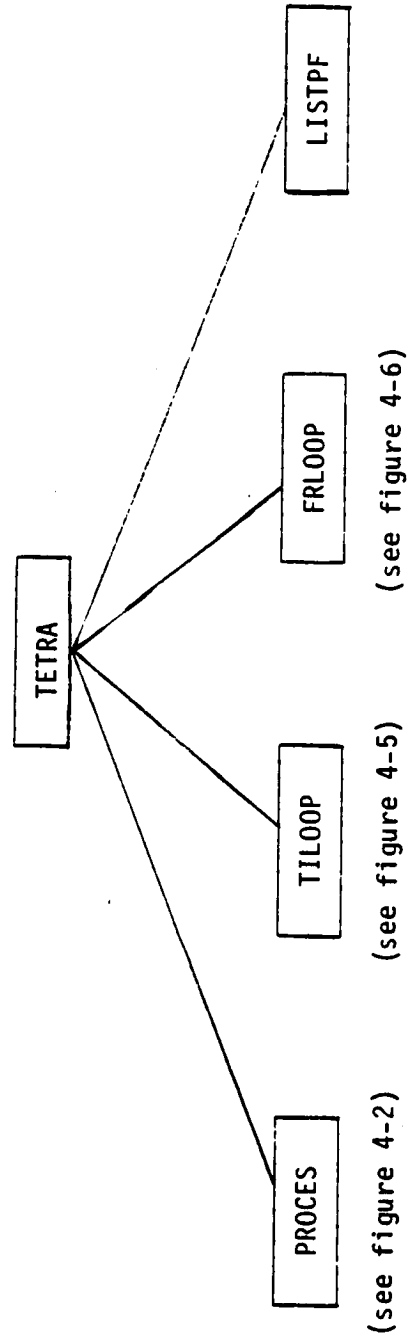
TETRA 2 also requires appreciably more memory than the previous version of TETRA. A major reason that it needs more memory is for the storage of the large global matrices. These global matrices are needed for the steady state analysis portion of TETRA 2, but were not needed in the previous version of TETRA which was limited to transient analyses. However, the amount of memory required by TETRA 2 (slightly over half a million words) is rather small compared to many structural analysis computer codes, such as those used for finite element analysis.

The structure of the TETRA 2 computer program is shown in figures 4-1 through 4-7. As is seen from figure 4-1, the main routine of TETRA calls four different subroutines: PROCES, TILOOP, FRLOOP, and LISTPF. Three of these subroutines, PROCESS, TILOOP, and FRLOOP, are the control routines for the three major branches of the program. The branch headed up by the PROCESS subroutine processes the input data, which is largely common to both transient and steady state analyses. The branch headed by the TILOOP subroutine is for the time integration loop, which is the solution phase for transient analysis runs. Finally, the branch headed up by the FRLOOP subroutine is for the frequency response loop, which is the solution phase for steady state analysis runs. Figures 4-2 through 4-7 show the branches of the program in detail. Included are all the TETRA 2 subroutines and all the IMSL library subroutines that are called directly from TETRA 2 subroutines. Omitted for clarity are those IMSL

subroutines that are not called directly from TETRA 2 subroutines but rather from the other IMSL library subroutines in TETRA 2.

Table 4-1 gives a brief description of each of the subroutines of TETRA 2, arranged alphabetically. Included are all the TETRA 2 subroutines and all the IMSL library subroutines that are called directly from TETRA 2 subroutines. Again, those IMSL subroutines that are not called directly from TETRA 2 subroutines, but rather from the other IMSL library subroutines in TETRA 2 are omitted.

Figure 4-1. TETRA 2 Structure Chart



TETRA - Executive routine for the program.

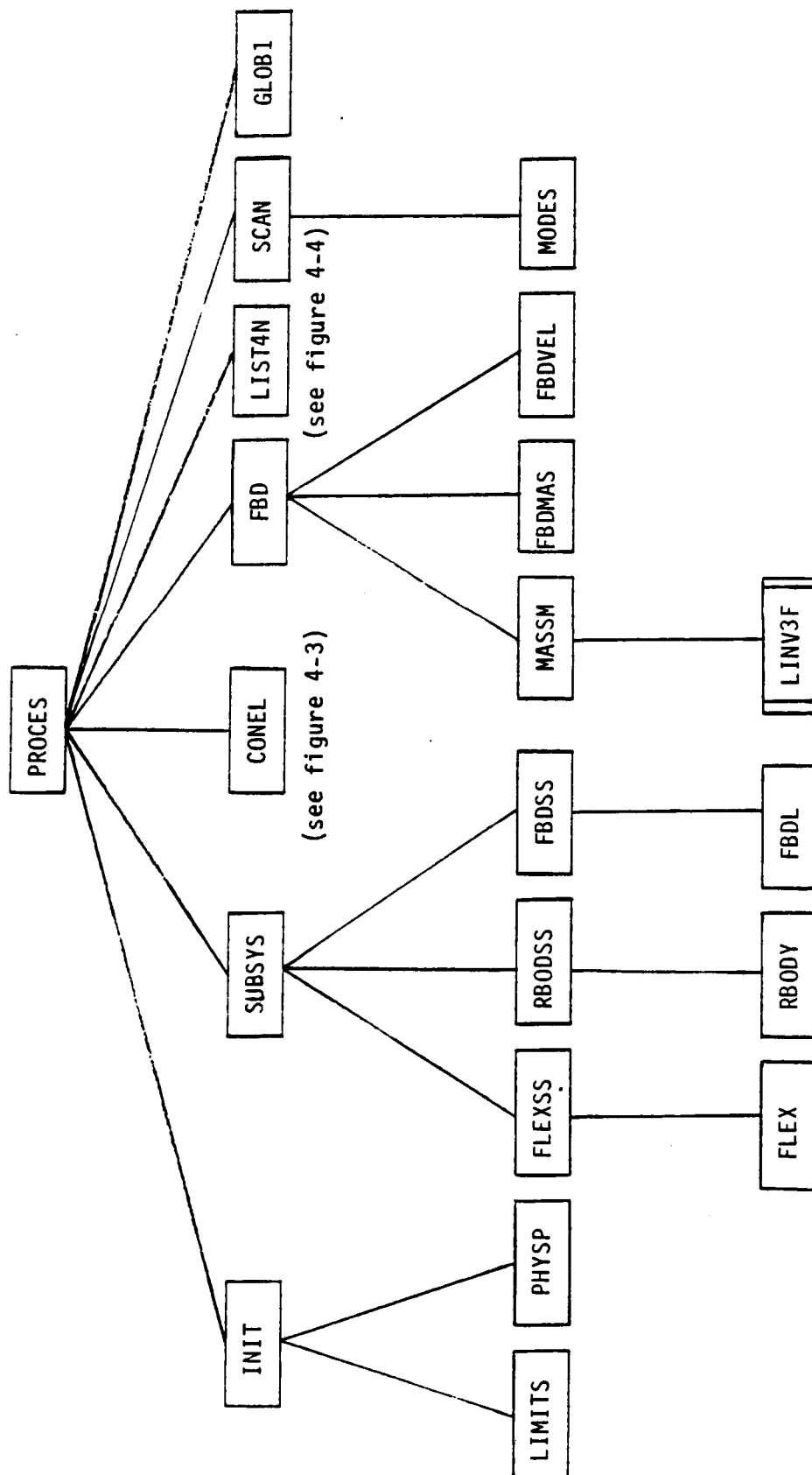
PROCES - Processes the input data.

TILLOOP - Control routine for the time integration loop (transient analysis solution branch).

FRLLOOP - Control routine for the forced response loop (steady state analysis solution branch).

LISTPF - Provided that a plot file was produced and that the long form of the output was requested, lists at least part of the output plot file.

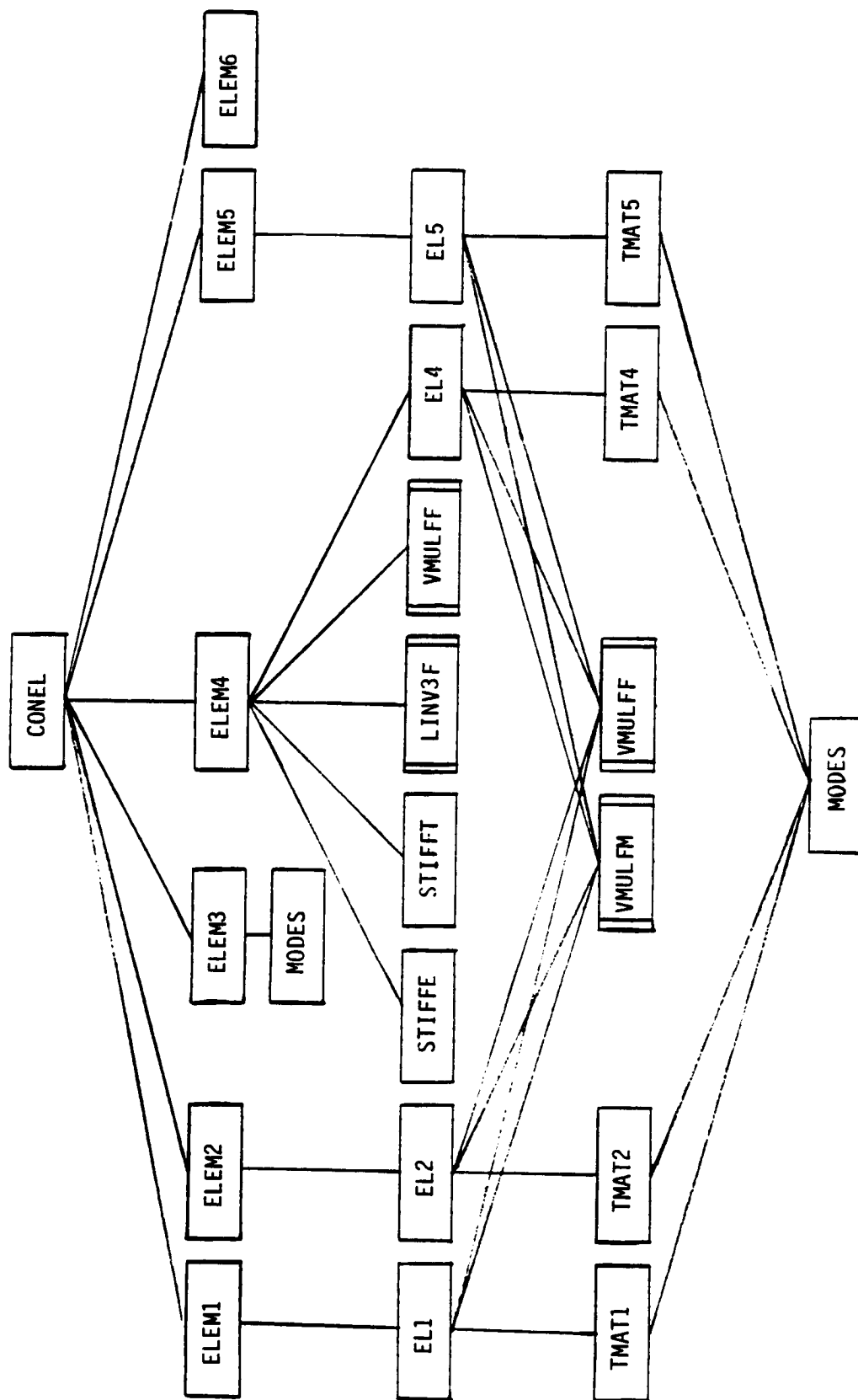
Figure 4-2. TETRA 2 Structure Chart (Continued)  
(Parent chart is figure 4-1)



denotes IMSL library subroutine



Figure 4-3. TETRA 2 Structure Chart (Continued)  
(Parent chart is figure 4-2)



 denotes IMSL library subroutines

Figure 4-4. TETRA 2 Structure Chart (Continued)  
 (Parent chart is figure 4-2)

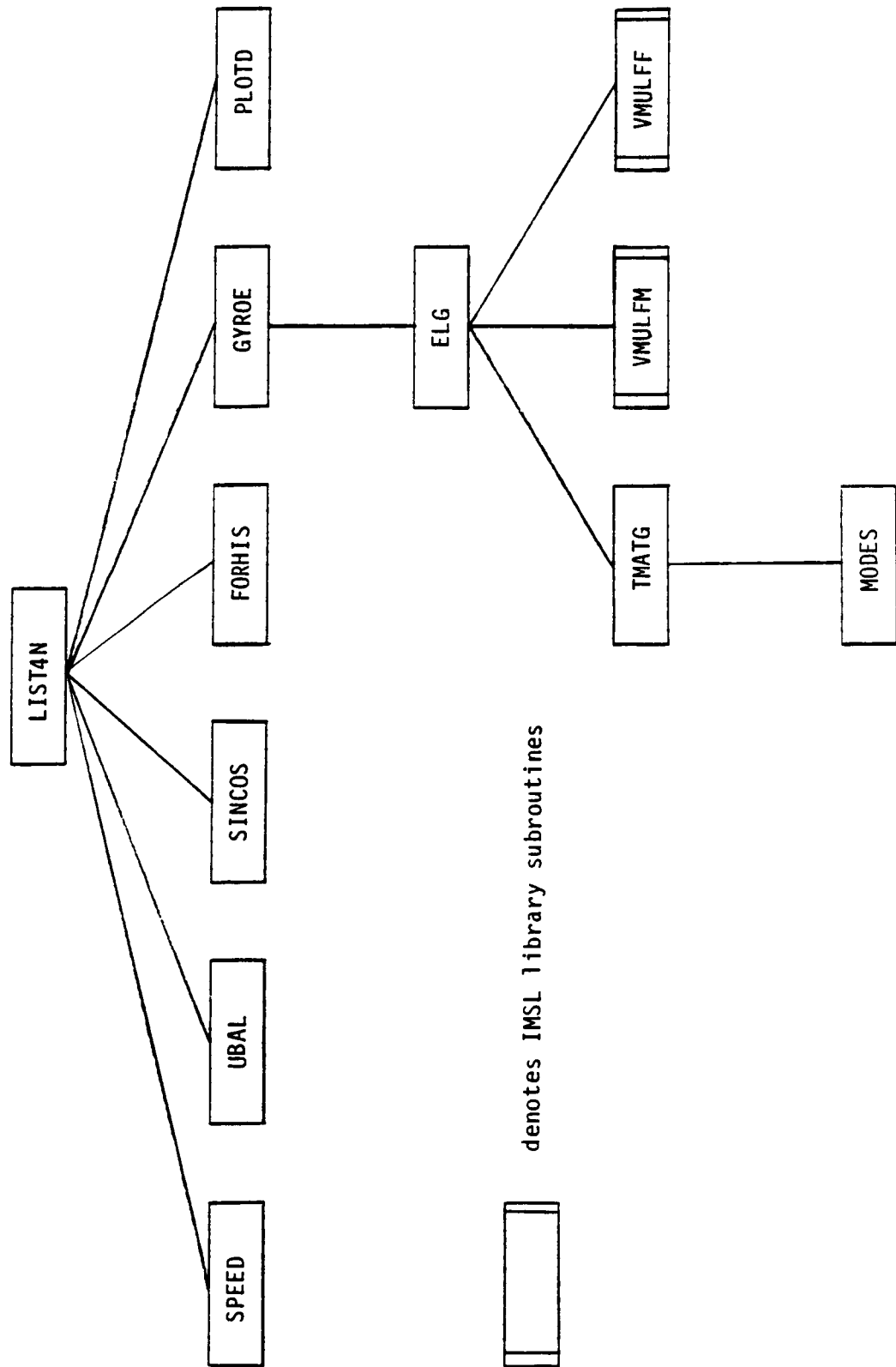


Figure 4-5. TEIRA 2 Structure Chart (Continued)  
 (Parent chart is figure 4-1)

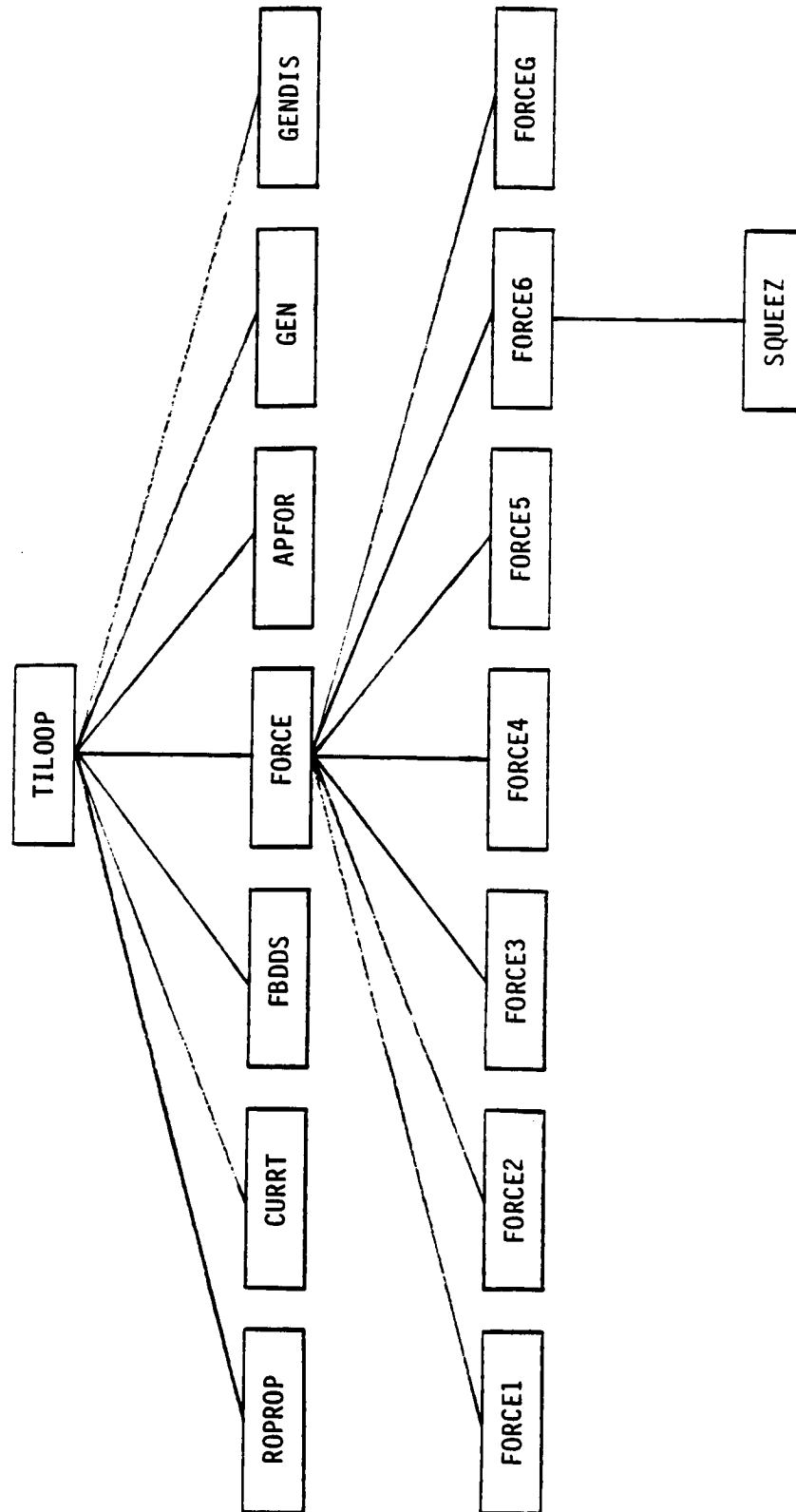
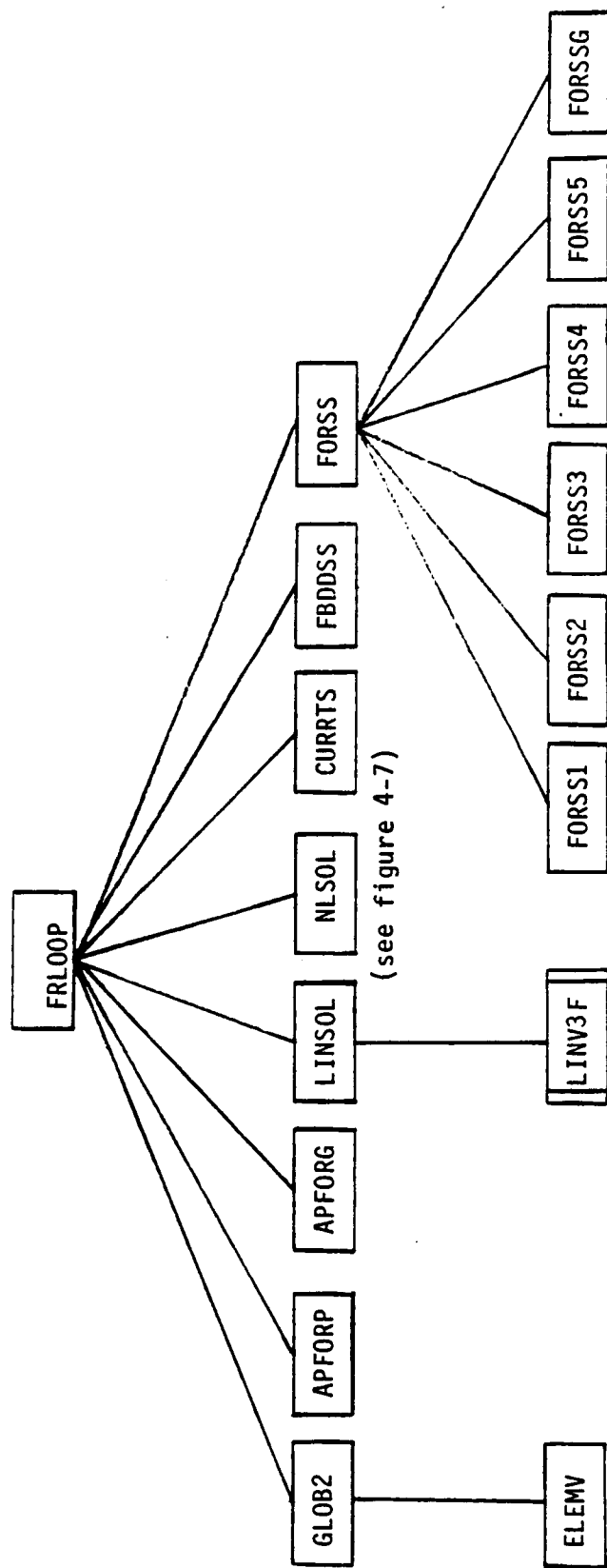
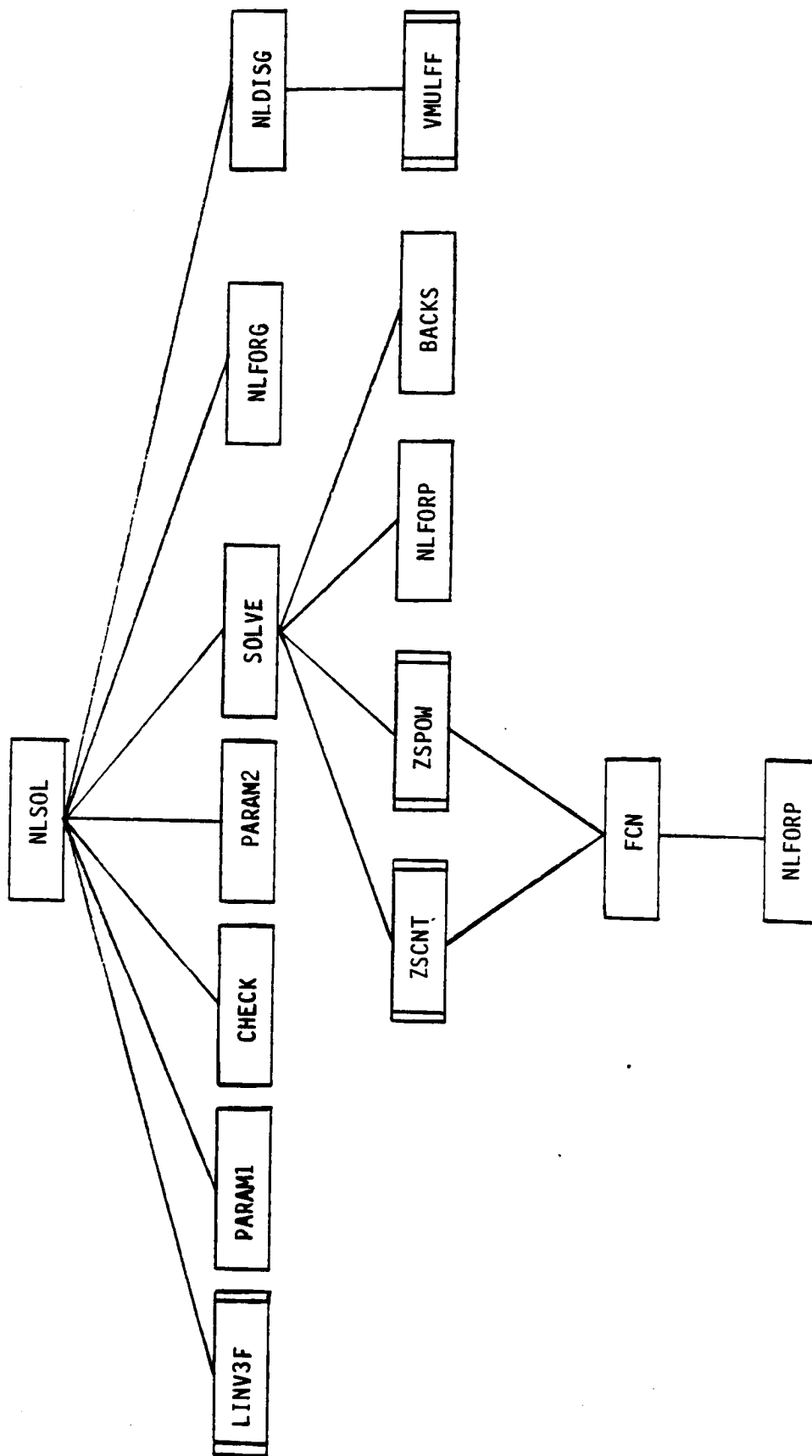


Figure 4-6. TETRA 2 Subroutine Chart (Continued)  
(Parent chart is Figure 4-1)



 denotes IMSL library subroutine

Figure 4-7. TETRA 2 Subroutine Chart (Continued)  
 (Parent chart is figure 4-6)



denotes IMSL subroutines

Table 4-1. TETRA 2 Subroutines

APFOR	Calculates the applied forces for transient analyses
APFORG	Calculates the applied generalized forces for steady state analyses
APFORP	Calculates the applied physical forces for steady state analyses
BACKS	Back substitutes the relative displacement components found by iteration into the iterating equations, to provide a measure of the quality of the solution (steady state analyses only)
CHECK	Checks if a rub is present for any of the physical connecting elements and acts accordingly (steady state analyses only)
CONEL	Control routine for the branch of the program which processes the input data for the physical connecting elements (transient and steady state analyses)
CURRT	Calculates the magnitudes of the physical displacements, physical velocities, and modal forces for transient analyses
CURRTS	Calculates the magnitudes and phase angles of the physical displacements, physical velocities, and modal forces for steady state analyses
EL1	Computes the contributions of the current type 1 physical connecting element (general spring-damper element) to the global stiffness matrix, the global velocity matrix, and the structural damping contributions matrix (steady state analyses only)
EL2	Computes the contributions of the current type 2 physical connecting element (space link-damper element) to the global stiffness matrix, the global velocity matrix, and the structural damping contributions matrix (steady state analyses only)
EL4	Computes the contributions of the current type 4 physical connecting element (engine support-links element) to the global stiffness matrix, the global velocity matrix, and the structural damping contributions matrix (steady state analyses only)
EL5	Computes the contributions of the current type 5 physical connecting element (uncoupled point spring-damper element) to the global stiffness matrix, the global velocity matrix, and the structural damping contributions matrix (steady state analyses only)
ELEM1	Processes the input data for the current type 1 physical connecting element (general spring-damper element) (transient and steady state analyses)
ELEM2	Processes the input data for the current type 2 physical connecting element (space link-damper element) (transient and steady state analyses)
ELEM3	Processes the input data for the current type 3 physical connecting element (rub element) (transient and steady state analyses)
ELEM4	Processes the input data for the current type 4 physical connecting element (engine support-links element) (transient and steady state analyses)
ELEM5	Processes the input data for the current type 5 physical connecting element (uncoupled point spring-damper element) (transient and steady state analyses)

ELEM6 Processes the input data for the current type 6 physical connecting element (squeeze film damper element) (transient analyses only)

ELEMV Finds the contributions of the linear (types 1, 2, 4, and 5) physical connecting elements (if any) and the gyroscopic elements (if any) to the upper right quadrant of the global solution matrix (steady state analyses only).

ELG Computes the contribution of the current gyroscopic element to the gyroscopic contributions matrix for the independent rotor (if the gyro element lies on the independent rotor) or to the gyroscopic element contributions matrix for the dependent rotor (if the gyro element lies on the dependent rotor) (steady state analyses only)

FBD Processes data for the flexible bladed disks (if any) (transient and steady state analyses)

FBDSS Calculates the magnitudes of the physical displacements and stresses of the local points on the flexible bladed disk(s) (if any) for transient analyses

FBDSSS Calculates the magnitudes of the physical displacements and stresses of the local points on the flexible bladed disk(s) (if any) for steady state analyses

FBDL Processes the input data for the local points on the flexible bladed disk(s) (if any) (transient and steady state analyses)

FBDMAS Computes the non-diagonal terms of the global mass matrix that result from the flexible bladed disk(s) (steady state analyses)

FBDSS Processes the input data for the flexible bladed disk subsystems (if any) (transient and steady state analyses)

FBDVEL Computes the non-diagonal terms of the global velocity matrix that result from the flexible bladed disk(s) (if any) (steady state analyses only)

FCN Computes the values of the iterating equations for steady state nonlinear analyses in which a rub has occurred

FLEX Processes the input mode shape data for flexible subsystems (transient and steady state analyses)

FLEXSS Processes the input data for the flexible subsystems (transient and steady state analyses)

FORCE Calculates the forces that the physical connecting elements and gyroscopic elements exert on the subsystems or ground for transient analyses

FORCE1 Calculates the forces that the type 1 physical connecting elements (general spring-damper elements) exert on the subsystems or ground for transient analyses

FORCE2 Calculates the forces that the type 2 physical connecting elements (space link-damper elements) exert on the subsystems or ground for transient analyses

FORCE3 Calculates the forces that the type 3 physical connecting elements (rub elements) exert on the subsystems for transient analyses

FORCE4 Calculates the forces that the type 4 physical connecting elements (engine support-links elements) exert on the subsystems or ground for transient analyses

FORCE5 Calculates the forces that the type 5 physical connecting elements (uncoupled point spring-damper elements) exert on the subsystems or

ground for transient analyses

FORCE6 Calculates the forces that the type 6 physical connecting elements (squeeze film damper elements) exert on the subsystems or ground for transient analyses

FORCEG Calculates the forces that the gyroscopic elements exert on the rotors for transient analyses

FORHIS Processes the input data for the time-force history loads (transient analyses only)

FORSS Calculates the forces that the physical connecting elements or gyroscopic elements exert on the modal subsystems or ground for steady state analyses

FORSS1 Calculates the forces that the type 1 physical connecting elements (general spring-damper elements) exert on the subsystems or ground for steady state analyses

FORSS2 Calculates the forces that the type 2 physical connecting elements (space link-damper elements) exert on the subsystems or ground for steady state analyses

FORSS3 Calculates the forces that the type 3 physical connecting elements (rub elements) exert on the subsystems for steady state analyses

FORSS4 Calculates the forces that the type 4 physical connecting elements (engine support-links elements) exert on the subsystems or ground for steady state analyses

FORSS5 Calculates the forces that the type 5 physical connecting elements (uncoupled point spring-damper elements) exert on the subsystems or ground for steady state analyses

FORSSG Calculates the forces that the gyroscopic elements exert on the rotors for steady state analyses

FRLOOP Loops over the forcing frequencies and finds the solution for each forcing frequency in turn. This subroutine is the control routine for the solution phase of steady state analyses.

GEN Calculates the generalized forces for transient analyses

GENDIS Calculates the generalized displacements for transient analyses

GLOB1 Prints out (provided that the user requested the long form of the printout) the global stiffness matrix, the global velocity matrix, the structural damping contributions matrix, the independent rotor gyroscopic contributions matrix, and the dependent rotor gyroscopic contributions matrix (steady state analyses only)

GLOB2 Completes the assembly of the global solution matrix, including the terms that vary with rotor speed of forcing frequency (steady state analyses only)

GYROE Processes the gyroscopic load input (transient and steady state analyses)

INIT Performs certain initial tasks, namely: initializing, then reading the LIST1 namelist input, printing out the title page, listing the input file while at the same time counting the number of subsystems NSUB and the number of physical connecting elements NELEK, listing the program limits, processing the data for points not on the modal subsystems, and (for steady state analyses) initializing several matrices to 0 (transient and steady state analyses)

LIMITS Prints the current program limits for the input variables (transient



and steady state analyses)

LINSOL Solves for the generalized displacements for a linear steady state analysis (one for which there are no nonlinear physical connecting elements)

LIST4N Processes the LIST4 namelist input (miscellaneous input that comes after the LIST1, LIST2, and LIST3 namelist input sections)

LISTPF Lists at least a portion of the output plot file (provided that an output plot file was produced and that the user requested the long form of the printed output)

MASSM Computes the mass matrix for the rotor on which the flexible bladed disk(s) are located and for the flexible bladed disk(s) (transient analyses only)

MODES Finds the mode shape given the modal subsystem number, local mode number, local point number, and local direction number (transient and steady state analyses)

NLDISG Calculates the generalized displacements for steady state analyses in which at least one nonlinear (rub) element is present

NLFORG Calculates the generalized forces for the type 3 physical connecting elements (rub elements) for steady state nonlinear analyses in which a rub has occurred

NLFORP Calculates the physical forces at joint I for each type 3 physical connecting element (rub element) for steady state analyses in which a rub has occurred

NLSOL Solves for the generalized displacements for steady state analyses that include nonlinear type 3 physical connecting elements (rub elements)

PARAM1 Finds the S parameters for steady state analyses in which at least one type 3 physical connecting element (rub element) is present. These parameters are needed to check if a rub has occurred, and, if a rub has occurred, then these parameters are also needed for use in the iterating equations.

PARAM2 Finds the A, B, C, and D parameters for steady state analyses in which a rub has occurred. These parameters are needed for use in the iterating equations.

PHYSP Processes input data for the physical points not located on the modal subsystems (transient and steady state analyses)

PLOTD Processes the input plot data and, if a plot file was requested, writes the initial plot file data onto the output plot file (transient and steady state analyses)

PROCES Processes the input data (transient and steady state analyses)

RBODSS Processes the input data for the rigid body modal subsystems (transient and steady state analyses)

RBODY Processes the mode shape input data for the rigid body modal subsystems (transient and steady state analyses)

ROPROP Calculates the rotor properties (rotor speed, acceleration, and angular displacement for the independent and dependent rotors) (transient analyses only)

SCAN Scans over the subsystems, points, directions, and modes and saves data needed later to calculate the generalized forces (transient and steady state analyses) and the physical displacements, physical velocities, and modal forces (transient analyses only)

SINCOS Processes the input data for the transient analysis  $P\cos(\omega t)$  and  $P\sin(\omega t)$  loads and the steady state analysis  $P\cos(\omega t + \Phi)$  loads  
 SOLVE Solves the nonlinear equations (via iteration) for steady state analyses in which a rub has occurred  
 SPEED Processes the rotor speed and rate input (transient and steady state analyses)  
 SQUEEZ Calculates the forces that the current type 6 physical connecting element (squeeze film damper element) exerts on the modal subsystems or ground for transient analyses  
 STIFFE Computes the engine support-links element stiffness matrix (transient and steady state analyses)  
 STIFFT Computes the truss (link) stiffness matrix for the trusses (links) that are part of the engine support-links element  
 SUBSYS Processes the input data for the modal subsystems (transient and steady state analyses)  
 TILLOOP Subroutine for the time integration loop, which finds the solution for each time step. This subroutine is the control loop for the solution phase for transient analyses.  
 TMAT1 Finds the transformation matrix and associated variables for the current type 1 physical connecting element (general spring-damper element) (steady state analyses only)  
 TMAT2 Finds the transformation matrix and associated variables for the current type 2 physical connecting element (space link-damper element) (steady state analyses only)  
 TMAT4 Finds the transformation matrix and associated variables for the current type 4 physical connecting element (engine support-links element) (steady state analyses only)  
 TMAT5 Finds the transformation matrix and associated variables for the current type 5 physical connecting element (uncoupled point spring-damper element) (steady state analyses only)  
 TMATG Finds the transformation matrix and associated variables for the current gyroscopic element (steady state analyses only)  
 UBAL Processes the input unbalance loads (transient and steady state analyses)

## 5.0 STEADY STATE ANALYSIS VERIFICATION

Verification of the steady state option of TETRA 2 is achieved at different levels of complexity. This is done partly to check the logic of the problem formulation and partly to demonstrate the correctness of the coding.

The initial stage consisted of checking degenerate cases which have been available in the literature for some time. These are usually linear single or two degree-of-freedom systems. TETRA 2 results agreed exactly with the closed form solutions for these cases (see section 5.1).

Next, a solution of Duffing's equation for the nonlinear hardening (or softening) spring (rub element) is included (see section 5.2.1). Also, a more complex system of a rigid beam with two nonlinear bearings (rub elements) was analyzed (see section 5.2.2). For both of these cases, TETRA 2 produced the same results as small computer programs which were independently programmed.

The final test cases use the two component demonstrator model consisting of a casing and a rotor. These cases are numerical. Frequency sweep results for the demonstrator model without any nonlinear springs (rub elements) were compared to the results of the NOVAS computer program (a GE program for engine vibrations analysis) (see section 5.3.2).

The demonstrator model including a nonlinear spring (rub element) was also analyzed. These results could not be checked against another computer program, since we currently do not have any other computer program with this capability. However, we did compare the results of the TETRA 2 transient analysis option when taken to steady state conditions with the direct solution obtained in the steady state option for these cases. This was done for each of the four possible rub categories: no rub, continual rub with dead band equal to zero, continual rub with dead band not equal to zero, and intermittent rub (see section 5.3.3).

These test cases have verified the logic and the coding of TETRA 2, specifically the steady state option. However, the transient analysis option was verified also by comparing its results with the original TETRA program.

## 5.1 Degenerate Cases: Closed Form Solutions

These cases are the classic spring-mass-dashpot systems in one and two degrees of freedom. Formulae from references 4 and 5 were programmed independently and compared to TETRA 2 results for the simplified models.

### 5.1.1 Single Degree-of-Freedom

A schematic of the physical model is shown in figure 5-1 and is self explanatory. The closed form equations were programmed independently on an IBM PC, and the resulting displacements tabulated in figure 5-1. The results using TETRA 2 are shown in the displacement versus frequency plot (figure 5-2). As can be seen by comparing the tabulated results in figure 5-1 with the plot of figure 5-2, exact agreement between TETRA 2 and the closed form equations was obtained.

### 5.1.2 Two Degree-of-Freedom

Figure 5-3 illustrates the other linear degenerate case, where again the resulting displacements tabulated in figure 5-3 were obtained by programming formulae obtained from references 4 and 5 on an IBM PC. TETRA 2 results for the same case are given in figures 5-4 (displacement versus frequency plot for point 1) and in figure 5-5 (displacement versus frequency plot for point 2). Again, as can be seen by comparing the tabulated results of figure 5-3 with the plots of figures 5-4 and 5-5, exact agreement between TETRA 2 and the closed form equations was obtained.

C:\TETRA>CFORM1

ENTER F0 (LB), K (LB/IN), C ((LB-SEC)/IN), AND M (LB)  
1000,5E5,87.8,500

ENTER BEGINNING, FINAL, AND STEP W (RPM)  
1000,20000,1000

FORCING FREQUENCY $\omega$ RPM	DISPLACEMENT AMPLITUDE Y INCHES
--------------------------------------	---------------------------------------

1000.	.00205805
2000.	.00225418
3000.	.00267875
4000.	.00363078
5000.	.00656400
6000.	.01778637
7000.	.00486259
8000.	.00241110
9000.	.00152751
10000.	.00108271
11000.	.00081886
12000.	.00064627
13000.	.00052579
14000.	.00043765
15000.	.00037087
16000.	.00031886
17000.	.00027744
18000.	.00024384
19000.	.00021617
20000.	.00019307

Stop - Program terminated.

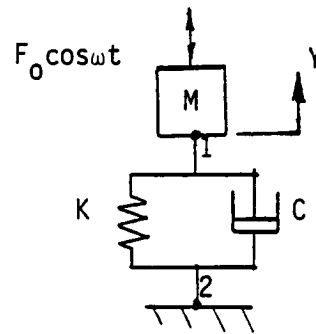


Figure 5-1. Closed Form Equation Results for 1 Degree-of-Freedom Linear Test Case

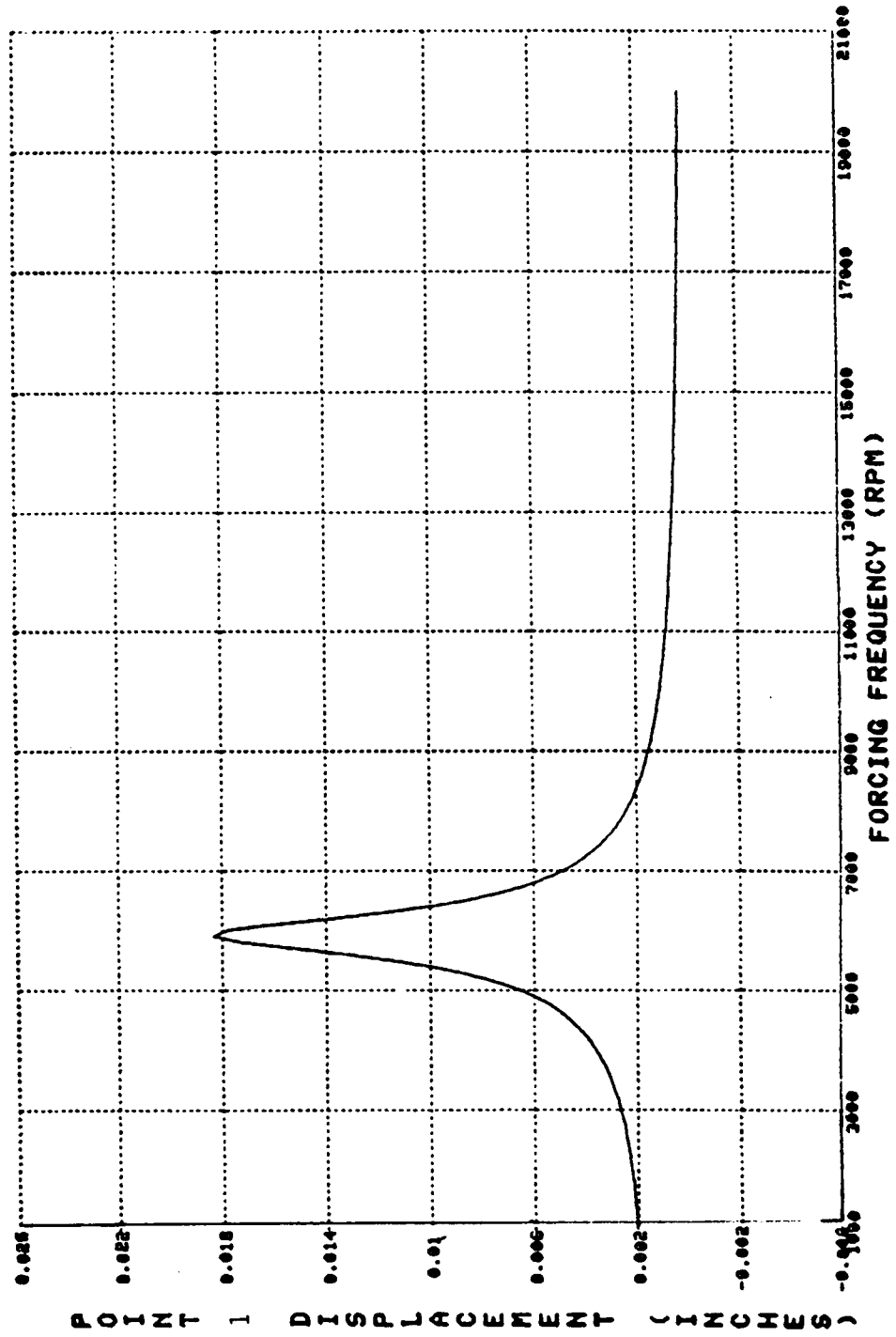


Figure 5-2. TETRA 2 Results for 1 Degree-of-Freedom Linear Test Case

C:\TETRA>CFORM2

ENTER F0 (LB), K1 (LB/IN), K2 (LB/IN), C2 ((LB-SEC)/IN),  
M1 (LB), AND M2 (LB)  
500,6E5,5E5,90,1000,100

ENTER BEGINNING, FINAL, AND STEP W (RPM)  
1000,20000,1000

FORCING FREQUENCY $\omega$ RPM	DISPLACEMENT AMPLITUDE Y1 INCHES	DISPLACEMENT AMPLITUDE Y2 INCHES
1000.	.00087910	.00088411
2000.	.00105294	.00107736
3000.	.00157399	.00165844
4000.	.00520835	.00572521
5000.	.00260335	.00302909
6000.	.00090976	.00113924
7000.	.00050937	.00070014
8000.	.00033416	.00051645
9000.	.00023725	.00042497
10000.	.00017617	.00038006
11000.	.00013451	.00036573
12000.	.00010631	.00037492
13000.	.00009426	.00039070
14000.	.00009849	.00035507
15000.	.00009625	.00025264
16000.	.00008509	.00016328
17000.	.00007359	.00010792
18000.	.00006399	.00007466
19000.	.00005619	.00005384
20000.	.00004981	.00004017

Stop - Program terminated.

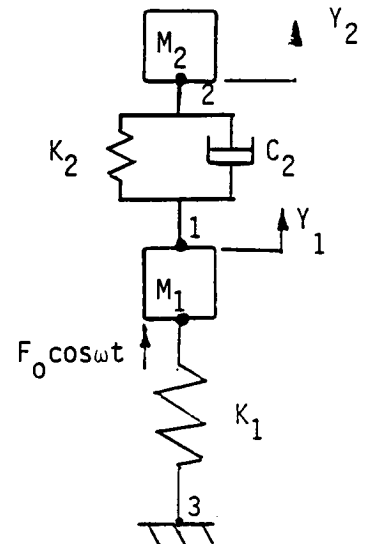


Figure 5-3. Closed Form Equation Results for 2 Degree-of-Freedom Linear Test Case

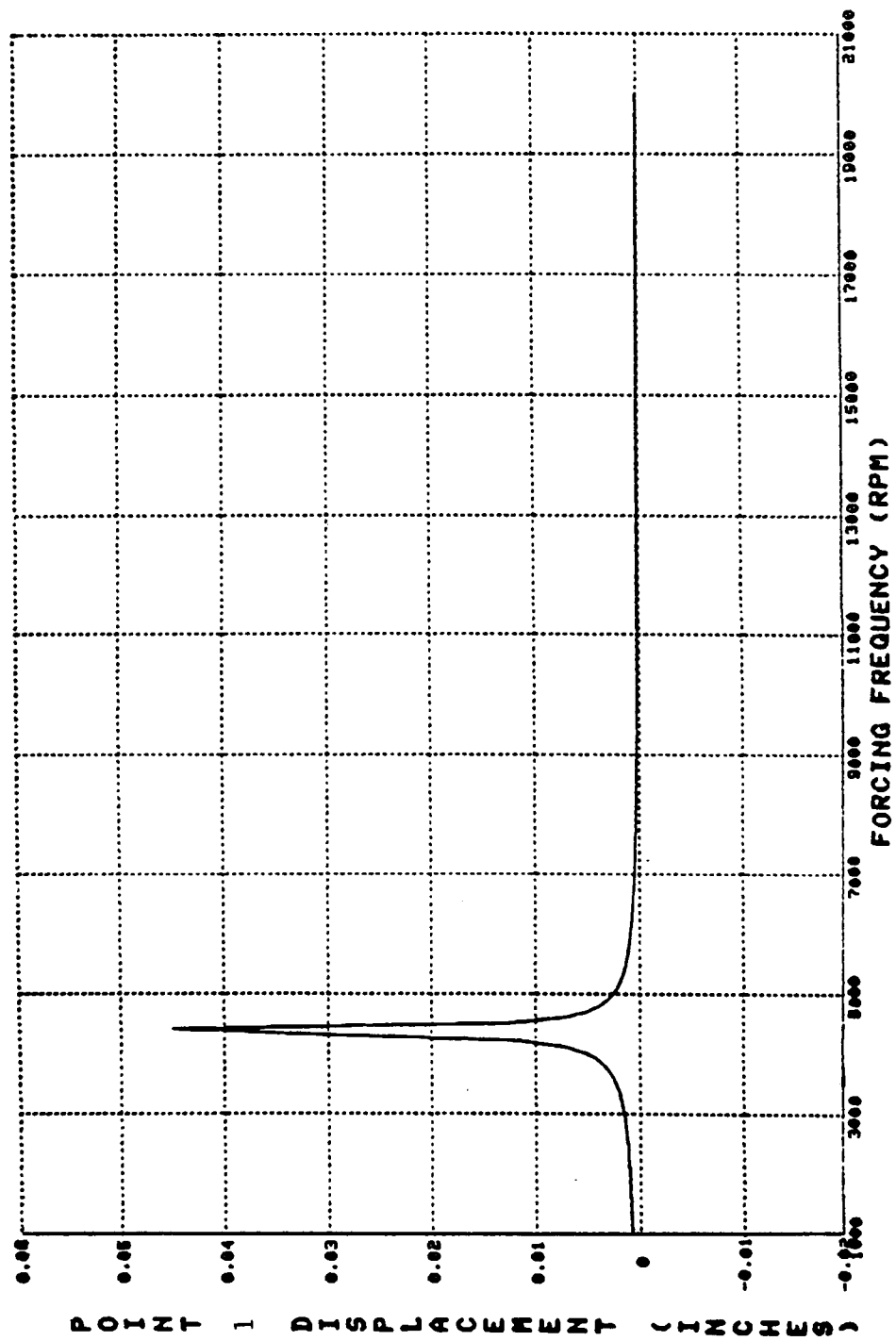


Figure 5-4. TETRA 2 Results for 2 Degree-of-Freedom Linear Test Case



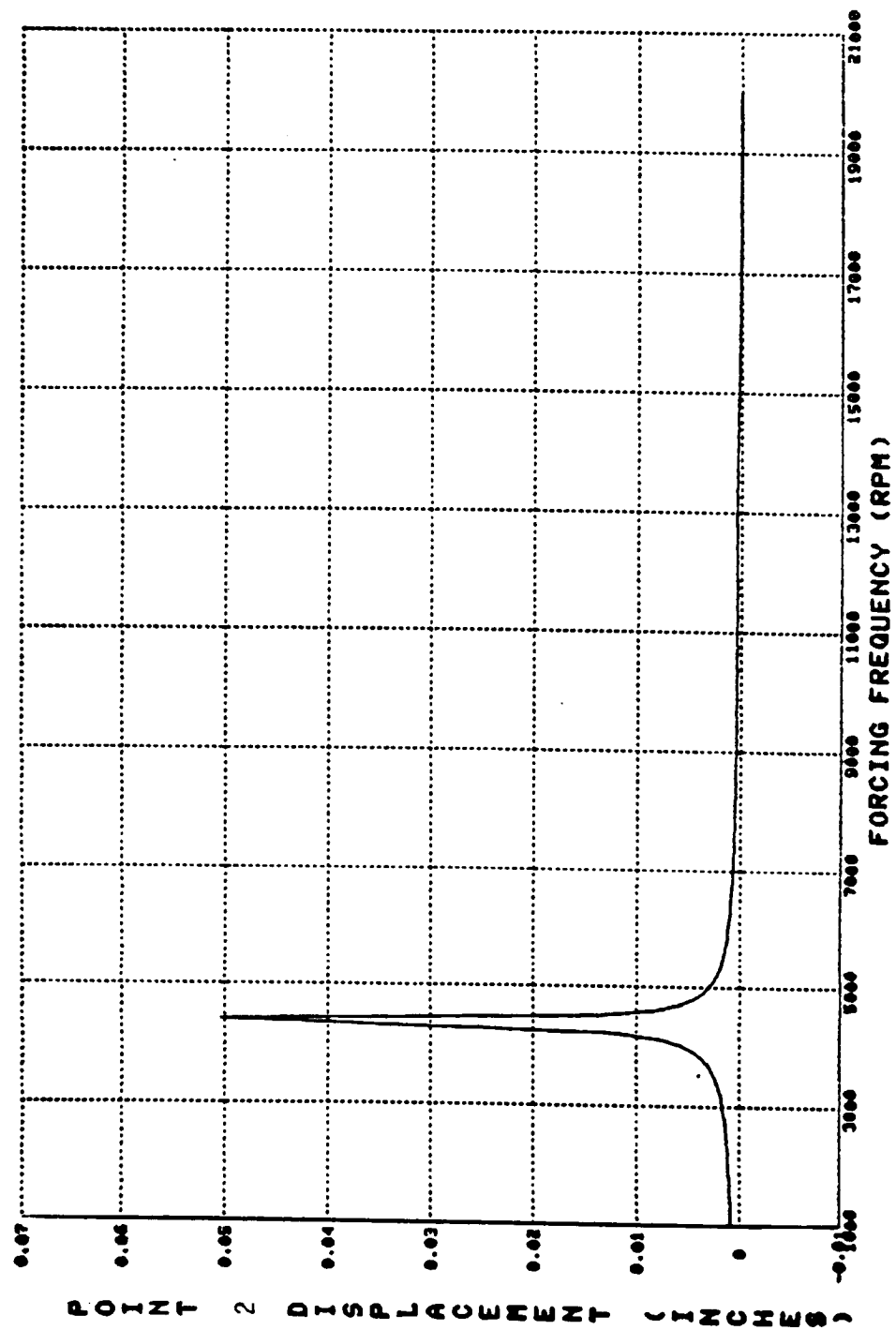


Figure 5-5. TETRA 2 Results For 2 Degree-of-Freedom Linear Test Case

## 5.2 Nonlinear Systems: Simple Models

To assess the iterative procedure for obtaining numerical solutions as well as check the logical method for determining the compatibility relations at the nonlinear connections, small computer programs were written to analyze the following two cases. Iterative subroutines from the IMSL library were initially used in these small programs and subsequently employed in TETRA 2.

Another purpose of writing the small computer programs cases was to obtain an independent check of the coding in TETRA 2. The people who programmed these small computer programs were not involved in the development or coding of TETRA 2. When TETRA 2 was run for these test cases, however, the same results as for the small computer programs were obtained. This provides confidence that both the small computer programs and the very much larger TETRA 2 were coded correctly.

The nonlinearity in the following cases is of the Duffing type, i.e. cubic hardening or softening spring with a linear viscous damper. This type of nonlinearity is discussed in references 4 and 5.

### 5.2.1 Single Degree-of-Freedom: Nonlinear Spring

This is the classic Duffing equation with a cubic hardening spring. A sketch of the model and the governing equations for this case are shown on the top part of figure 5-6.

A small computer program was written specifically to assess the IMSL library subroutine ZSYSTEM, which uses Brown's iteration method. Results of this effort were discussed in the monthly progress reports. The most salient conclusion is that this subroutine (and also IMSL subroutines ZSCNT and ZSPOW) can yield multiple solutions corresponding to the classic "jump" solution of Duffing's equation with large nonlinearity. However, to obtain the solutions in the neighborhood (in frequency) of the "jumps", one must run the program twice, once with increasing forcing frequency and once with decreasing forcing frequency. This is illustrated by the plot of this case shown in figure 5-6.

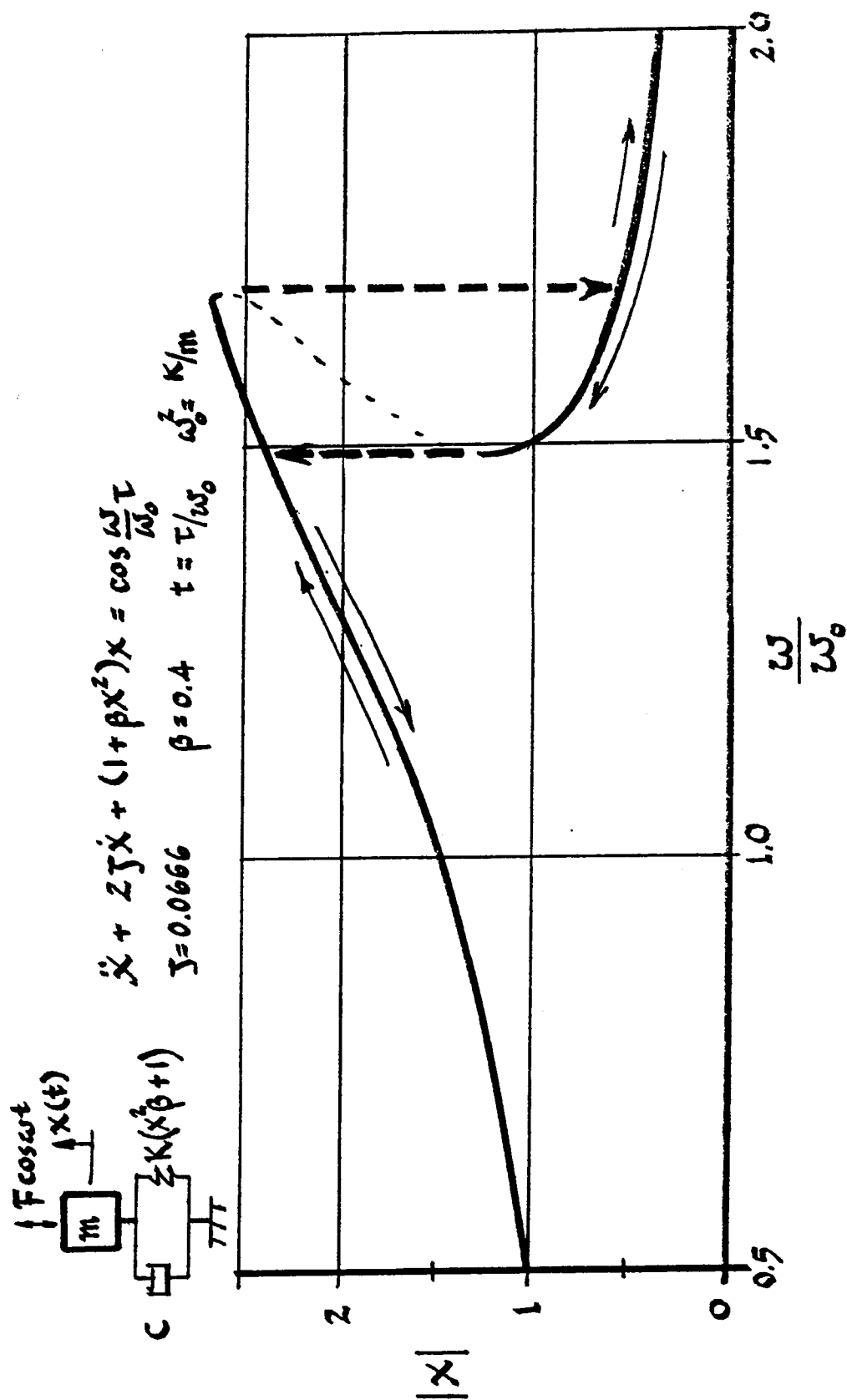
This procedure indicates the sensitivity of the iteration routine to initial trial solutions. In addition, the use of harmonic balance in the solution of Duffing's equation gives similar results as those obtained by perturbation or direct iteration (without harmonic balance).

TETRA 2 was also originally coded to include IMSL subroutine ZSYSTEM. The results produced by the small computer program and TETRA 2 were in excellent agreement. Soon after this case was run, however, IMSL Incorporated deleted subroutine ZSYSTEM from their library and replaced it with two new subroutines, ZSCNT and ZSPOW. The new iteration subroutines are based on the secant method and Powell's hybrid algorithm (a variation of Newton's method) respectively.

Subsequent test cases that we ran confirmed that ZSCNT and ZSPOW both worked better than subroutine ZSYSTEM, so we also deleted ZSYSTEM from TETRA 2 and

instead included ZSCNT and ZSPOW. Subroutine ZSYSTEM would frequently be unable to converge due to a singularity in the Jacobian, which was a great nuisance. This problem is not present in the ZSCNT or ZSPOW IMSL subroutines. All three IMSL subroutines give the same results when they converge.

Figure 5-6. Single Degree-of-Freedom With Hardening Spring (Duffing's Equation)



### 5.2.2 Planar Rigid Beam with Two Nonlinear Bearings

This model is the simplest one that checks TETRA 2's method for obtaining the iterating equations, i.e., displacement compatibility at the nonlinear bearings (rub elements). A sketch of this model is shown in figure 5-7.

The governing equations were obtained two different ways:

- (1) In terms of translation and rotation of the center of gravity
- (2) In terms of two rigid modes: the translation modal distribution simulates the cases of pure translation and simple rotation about the center of gravity.

Both formulations gave identical results. The responses that were compared to the TETRA 2 output were the physical displacement amplitudes at the bearings (rub elements). The first formulation is summarized here. Following volume 1 of this report, the generalized coordinate differential equations of motion are:

Translation:

$$m\ddot{y} + (K_1 + K_2)y + (C_1 + C_2)\dot{y} + (K_2L_2 - K_1L_1)\theta + (C_2L_2 - C_1L_1)\dot{\theta} = \bar{F}_y(t) - \mu_1 K_1 y_1^3 - \mu_2 K_2 y_2^3$$

Rotation about the center of gravity:

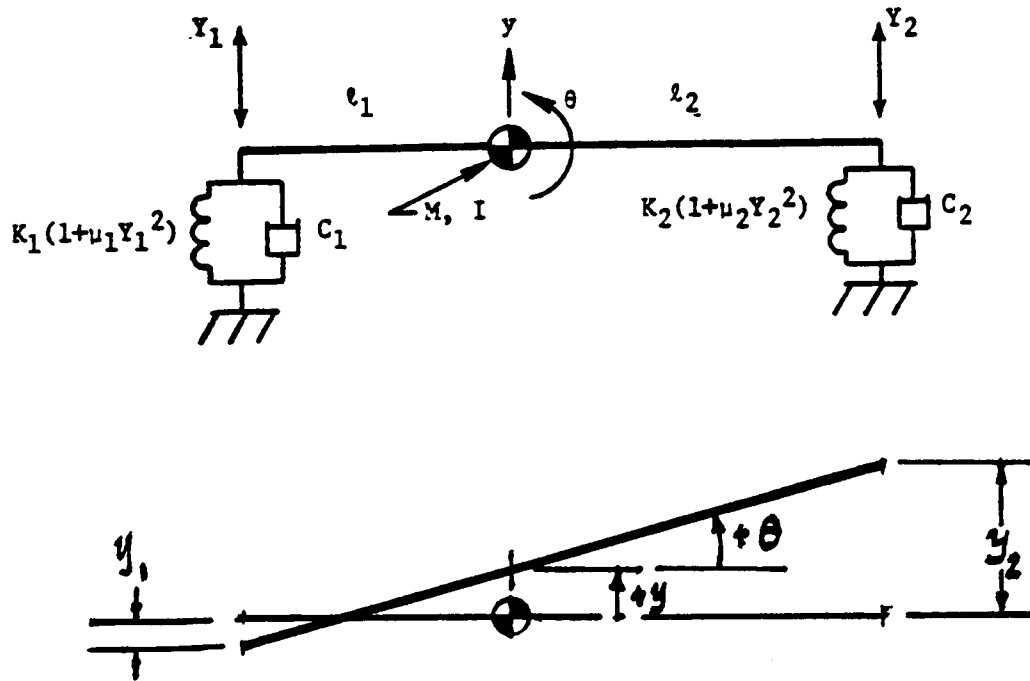
$$I\ddot{\theta} + (K_1L_1^2 + K_2L_2^2)\theta + (C_1L_1^2 + C_2L_2^2)\dot{\theta} + (K_2L_2 - K_1L_1)y + (C_2L_2 - C_1L_1)\dot{y} = \bar{F}_\theta(t) - \mu_1 K_1 L_1 y_1^3 + \mu_2 K_2 L_2 y_2^3$$

The symbols are defined in the sketch in figure 5-7.

These differential equations are transformed to 4 simultaneous nonlinear equations by the principle of harmonic balance following the theory of volume 1. After some simplification, these become a matrix relating the generalized coordinates of the center of gravity on the left hand side to the physical displacements and external forces on the right. Thus,

$$\begin{bmatrix} 1-v^2 & \alpha_{y\theta}^2 & 2v\zeta_y & 2v\zeta_{y\theta} \\ \alpha_{\theta y}^2 & p^2-v^2 & 2vp\zeta_{y\theta} & 2vp\zeta_\theta \\ -2v\zeta_y & -2v\zeta_{y\theta} & 1-v^2 & \alpha_{y\theta}^2 \\ -2vp\zeta_{y\theta} & -2vp\zeta_\theta & \alpha_{\theta y}^2 & p^2-v^2 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ b_1 \\ b_2 \end{bmatrix} = - \begin{bmatrix} \frac{3}{4}\mu_1 A_1(A_1^2+B_1^2) \\ + \frac{3}{4}\mu_2 A_2(A_2^2+B_2^2) \\ \frac{3}{4}p^2\mu_1 A_1(A_1^2+B_1^2) \\ - \frac{3}{4}p^2\mu_2 A_2(A_2^2+B_2^2) \\ \frac{3}{4}\mu_1 B_1(A_1^2+B_1^2) \\ + \frac{3}{4}\mu_2 B_2(A_2^2+B_2^2) \\ \frac{3}{4}p^2\mu_1 B_1(A_1^2+B_1^2) \\ - \frac{3}{4}p^2\mu_2 B_2(A_2^2+B_2^2) \end{bmatrix} + \begin{bmatrix} \frac{f_1}{\omega_y^2} \\ \frac{f_\theta}{\omega_y^2} \\ 0 \\ 0 \end{bmatrix}$$

Figure 5-7. Planar Rigid Beam with 2 Nonlinear Bearings



$$y_1 = y - l_1 \theta$$

$$y_2 = y + l_2 \theta$$

$$y = a_1 \cos \omega t + b_1 \sin \omega t; \quad \theta = a_2 \cos \omega t + b_2 \sin \omega t$$

$$y_1 = A_1 \cos \omega t + B_1 \sin \omega t; \quad y_2 = A_2 \cos \omega t + B_2 \sin \omega t$$

where:

$$\begin{aligned}
 \nu &= \omega/\omega_y & \mathcal{J}_y &= c_y/c_{cy} \\
 \rho &= \omega/\omega_y & \mathcal{J}_0 &= c_0/c_{c0} \\
 \alpha_{y0} &= \omega_{y0}/\omega_y & \mathcal{J}_{y0} &= c_{y0}/c_{cy} \\
 \alpha_{0y} &= \omega_{0y}/\omega_y & \mathcal{J}_{0y} &= c_{0y}/c_{c0} \\
 \omega_y^2 &= K_{yy}/m & \omega_{y0}^2 &= K_{y0}/m \\
 \omega_0^2 &= K_{00}/I & \omega_{0y}^2 &= K_{0y}/I \\
 \\ 
 K_{yy} &= K_1 + K_2 & K_{y0} &= K_2 L_2 - K_1 L_1 = K_{0y} \\
 K_{00} &= K_1 L_1^2 + K_2 L_2^2 & C_{y0} &= C_2 L_2 - C_1 L_1 = C_{0y} \\
 C_{yy} &= C_1 + C_2 & C_{c0} &= 2 I \omega_0 \\
 C_{00} &= C_1 L_1^2 + C_2 L_2^2 & C_{cy} &= 2 m \omega_y \\
 \\ 
 \mu_{1y} &= \mu_1 K_1/m & \mu_{2y} &= \mu_2 K_2/m \\
 \mu_{10} &= \mu_1 K_1 L_1/I & \mu_{20} &= \mu_2 K_2 L_2/I \\
 \\ 
 f_y &= F_y/m & f_0 &= F_0/I
 \end{aligned}$$

The algebraic matrix equation is first transformed by matrix inversion to obtain an explicit expression for the generalized coordinate vector. Letting:

$$\begin{aligned}
 E_{ij} &= \text{Coefficient of } a_i, b_i \\
 \alpha_i &= a_i, a_2, b_1, b_2 \\
 H_i &= \text{Nonlinear Connecting Force Vector} \\
 \bar{F}_i &= \text{External Force Vector}
 \end{aligned}$$

The matrix equation can be written more simply:

$$E_{ij} \alpha_j = \bar{F}_i + H_i(A_1, B_1, A_2, B_2, \mu_{1y}, \mu_{2y}, \mu_{10}, \mu_{20} \dots)$$

Solving for  $\alpha_j$  gives:

$$\alpha_j = E_{ij}^{-1} \{ \bar{F}_i + H_i \}$$

The displacement (cosine and sine) components at the ends may now be expressed in terms of  $a_i$  and  $b_i$ , since:

$$A_1 = a_1 - a_2 L_1$$

$$B_1 = b_1 - b_2 L_1$$

$$A_2 = a_1 + a_2 L_2$$

$$B_2 = b_1 + b_2 L_2$$

Noting that  $a_1, a_2, b_1$ , and  $b_2$  are discrete values of  $\alpha_j$ , and that the  $\alpha_j$  are expressed as nonlinear functions of  $A_1, B_1$ , etc., and substituting these into the physical displacement expressions gives the iterating equations, thus:

$$A_1 = (E_{i1}^{-1} - L_1 E_{i2}^{-1})(\bar{F}_i + \mathcal{H}_i(A_1, B_1, A_2, B_2))$$

$$A_2 = (E_{i4}^{-1} + L_2 E_{i2}^{-1})(\bar{F}_i + \mathcal{H}_i)$$

$$B_1 = (E_{i3}^{-1} - L_1 E_{i4}^{-1})(\bar{F}_i + \mathcal{H}_i)$$

$$B_2 = (E_{i3}^{-1} + L_2 E_{i4}^{-1})(\bar{F}_i + \mathcal{H}_i)$$

The foregoing follows the essential elements of the theory which proceed from the formation of the connecting element forces to the formation of a global generalized matrix differential equation and to its transformation by harmonic balance. It demonstrates also the steps required to obtain the iterating equations from the physical displacement relations at the bearings (rub elements). Thus, the application of the theory can be checked, step by step.

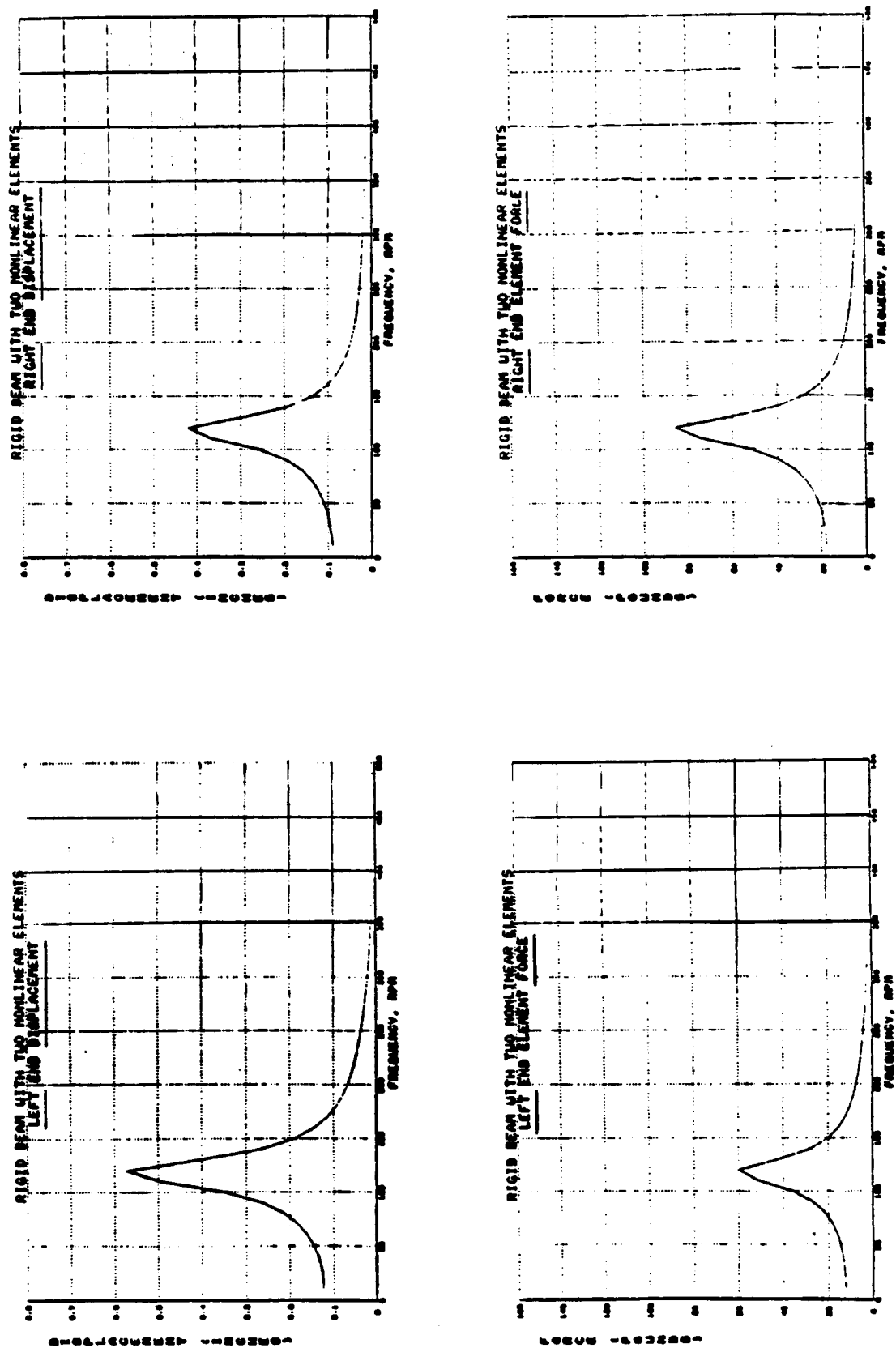
Finally, a small computer program was written to calculate the beam ends' response, and the results agreed with the TETRA 2 solution. This case also revealed potential convergence problems with the IMSL iteration subroutines. Large amplitudes (which amplify the nonlinearities) occasionally led to a large number of iterations that would not converge, especially in the regions of resonances. However, displacement amplitudes and nonlinear forces more typical of actual engine experience avoided the convergence problems.

For highly nonlinear connecting elements where a "jump" phenomenon is expected, manual inputs of initial iteration values in TETRA 2 (see input sheet I-2) can decrease the number of iterations and overcome non-convergence.

Figure 5-8 gives some response plots from TETRA 2 for this case.



Figure 5-8. Planar Rigid Beam with 2 Nonlinear Bearings



### 5.3 Demonstrator Model

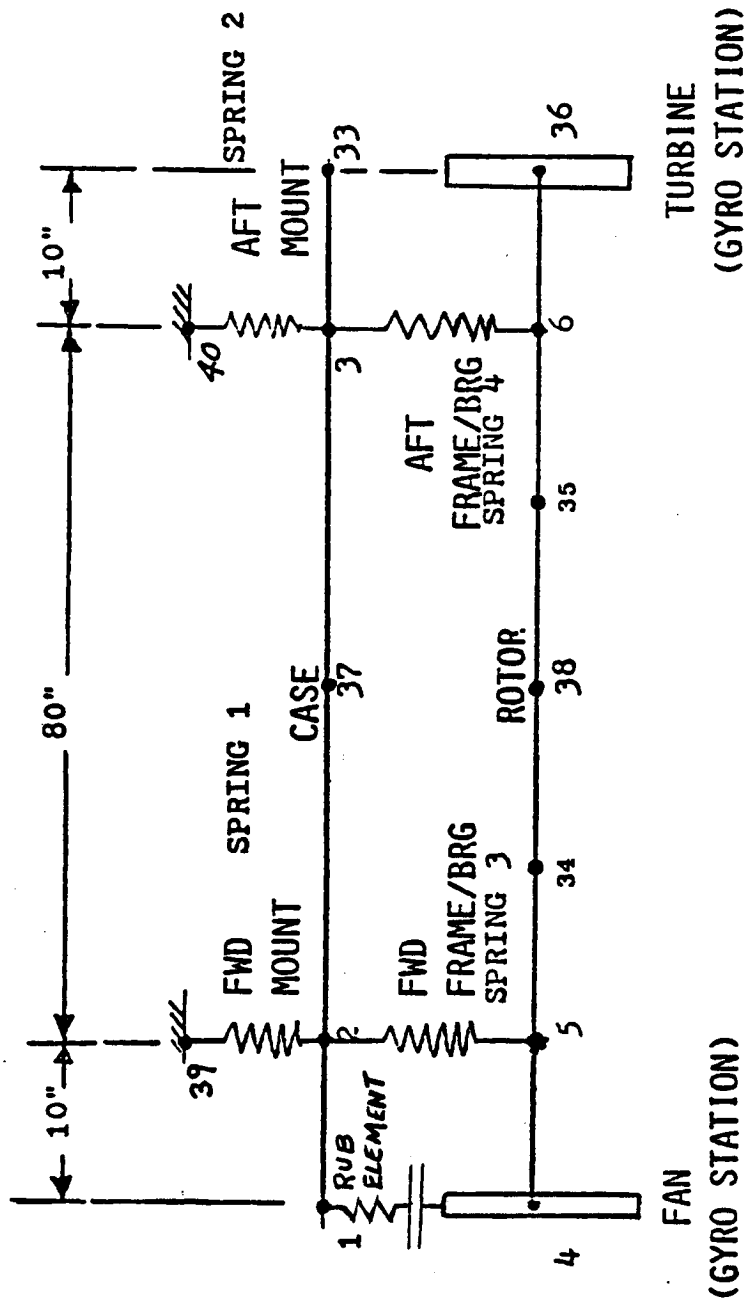
The final set of test cases use the two component demonstrator model consisting of a rotor and a casing. This is the exactly the same demonstrator model that was used back in 1981 to check out the original TETRA program (which was for transient analyses only), as documented in reference 1. For more details about this model, see reference 1 (especially volume 3, section 7.0). A sketch of the model is shown in figure 5-9.

This demonstrator model was also used in 1983 in modified form (two rotors, instead of a rotor and a casing, and the stiffnesses changed) to check out the flexible bladed disk option of TETRA (see reference 2). However, the test cases made to check out the steady state analysis portion of TETRA conform to the original version of the demonstrator model (rotor and casing), rather than the one that was modified to check out the flexible bladed disk option.

The earlier test cases for the demonstrator model (both with and without flexible bladed disk) were repeated, with the same results as before. This verifies that the extensive modifications made for TETRA 2 did not mess up the transient analysis portion of the program. The transient analysis portion of the program was improved with TETRA 2 by making it more efficient, and by making the new features developed for TETRA 2 (such as the improved rub element) applicable to both steady state and transient analyses.

In the following sections, we only concern ourselves with the new test cases that were made to verify the steady state analysis portion of TETRA 2. Reference 1 (volume 3) and reference 2 give details about the earlier test cases that verified the transient analysis portion of the program. To compare with the steady state analysis results, some new transient analysis runs were made which were taken to state analysis conditions (that is, run for a long enough time for the transient analysis part to die out), as reported in section 5.3.3.

Section 5.3.1 discusses the input and output for a steady state analysis test case for the demonstrator model. Then, in section 5.3.2 we compare frequency sweep results for the demonstrator without any nonlinear springs (rub elements) to the results of the NOVAS computer program (GE program for engine vibrations analysis). Finally, in section 5.3.3 we examine test cases for the demonstrator with one rub element, comparing steady state analysis results with transient analysis runs which were taken to steady state conditions. As will be seen, excellent results were obtained by these test cases, further verifying the new steady state analysis capability of TETRA 2.



#### PHYSICAL CONNECTING ELEMENTS

- Spring 1  $0.5 \times 10^6$  lb/in.
- Spring 2  $0.5 \times 10^6$  lb/in.
- Spring 3  $1.0 \times 10^6$  lb/in.
- Spring 4  $1.0 \times 10^6$  lb/in.

POINT 1 IS THE GLOBAL ORIGIN  
( $X=Y=Z=0$ ) FOR THE TETRA MODEL.  
PHYSICAL POINT NUMBERS 1,2,37,  
ETC. IDENTIFIED.

Figure 5-9. Demonstrator Model

### 5.3.1 Input and Output

TETRA 2 input and output follows for a steady state analysis run of the demonstrator model. The input listing is a part of the TETRA 2 output and immediately follows the title page of the output. This run corresponds to the run discussed in section 5.3.3.4 (intermittent rub at 3000 rpm). To produce this output, the standard form of the printed output was requested (IOUT = 1 on the type A input sheet).

```

TTTTTTTTT EEEEEEE TTTTTTTTT RRRRRRRR AAAAA
TTTTTTTTT EEEEEEE TTTTTTTTT RRRRRRRR AAAAA
TTTTTTTTT EEE EEE TTTTTTTTT RRR RRR AAA
TTT EEE EEE TTT TTT TTT AAA
TTT EEEEEEE TTT RRRRRRRR AAAAA
TTT EEEEEEE TTT TTT AAAAA
TTT EEE TTT TTT AAAAA
TTT EEEEEEE TTT RRR RRRR AAA
TTT EEEEEEE TTT RRR RRRR AAA
TTT EEEEEEE TTT RRR RRRR AAA

```

VERSION 2.0

G. BLACK DATE AUGUST 1980 CHARGE 19615  
 BLDG. 200 MAIL DROP G-60 EXT. 1129  
 ROTOR AND CASE VERTICAL AND HORIZONTAL MODEL 1  
 WITH GYRO LOADS AT FAN AND TURBINE

DATE 12/05/86  
 TIME 11:11:17

THIS RUN IS FOR A STEADY STATE ANALYSIS.  
 STRUCTURAL DAMPING, IF PRESENT, IS BASED ON  
 THE INDEPENDENT ROTOR SPEED.  
 THE STANDARD FORM OF THE OUTPUT IS BEING PRINTED.

LISTING OF INPUT FILE:

```

$LIST1
NAME='G. BLACK    DATE AUGUST 1980    CHARGE 19615'.
ADDRESS='BLDG. 200    MAIL DROP G-60    EXT. 1129'.
IDENT1='ROTOR AND CASE VERTICAL AND HORIZONTAL MODEL 1'.
IDENT2='WITH GYRO LOADS AT FAN AND TURBINE'.
NTYPE=1.
IOUT=1.
PP(1,1)=
39.1,-10.0,0.
40.1,-90.0,0.
$
$LIST2
TITLE='ROTOR VERTICAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1'.
ISUB=1.
PTS(1,1)=
4.0,0.0.
5.-10.0,0.
34.-30.0,0.
38.-50.0,0.
35.-70.0,0.
6.-90.0,0.
36.-100.0,0.
XMODES(1,1)=
99.3,192.0,1.
120.8,300.0,1.
10065.5,38E5,0.0.
16851.9,90E6,0.0.
26425.1,75E6,0.0.
VH(1,1,1)=
-1.-02.0,0.
-8.-02.0,0.
-4.-02.0,0.
0.-02.0,0.
-4.-02.0,0.
8.-02.0,0.
1.-02.0,0.
VH(1,1,2)=
1.0,0.0.
1.0,0.0.
1.0,0.0.
1.0,0.0.
1.0,0.0.
1.0,0.0.
VH(1,1,3)=
.12215,.034677,0.0.
-.25282,-.029534,-2.073E5,1.176E7.
-.80295,-.016102,-1.021E5,.568E7.
-1.0,3.519E4,1.711E7.
-.80295,-.016102,1.586E5,1.558E7.
-.25282,.029534,2.073E5,1.176E7.
.12215,.034677,0.0.
VH(1,1,4)=
.2748,.1075,0.0.
.8258,.0707,-1.444E6,-6.956E7.
.8584,.0174,-1.892E6,-3.877E7.
0.-.0017,-1.989E6,1.608E4.
-.8584,.0174,-1.723E6,3.846E7.

```

```

-.8258,.0707,-1.444E6,6.956E7,
-.2748,.1075,0.0,
VH(1,1,5)=
-.10361,-.01856,0.0,
.0959,-.00459,1.241E6,2.250E7,
.6966,.00594,6.883E6,-2.488E5,
1.0,-2.426E5,-9.743E6,
.6966,-.00594,-1.026E6,1.768E4,
.0959,.00459,-1.241E6,2.250E7,
-.10361,.01856,0.0,
$
$LIST2
TITLE='ROTOR HORIZONTAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1',
ISUB=2,
PTS(1,1)=
4.0,0.0,
5.-10.0,0.0,
34.-30.0,0.0,
38.-50.0,0.0,
35.-70.0,0.0,
6.-90.0,0.0,
36.-100.0,0.0,
XMODES(1,1)=
99.3,192.0,1,
120.8,300.0,1,
10065.5,38E5,0.0,
16851.9,90E6,0.0,
26425.1,75E6,0.0,
VH(1,1,1)=
-1.-02,0.0,
-.8.-02,0.0,
-.4.-02,0.0,
0.-02,0.0,
.4.-02,0.0,
.8.-02,0.0,
1.-02,0.0,
VH(1,1,2)=
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
VH(1,1,3)=
.12215,-.034677,0.0,
-.25282,-.029534,-2.073E5,1.176E7,
-.80295,-.016102,-1.021E5,1.568E7,
-1.0,3.519E4,1.711E7,
-.80295,.016102,1.586E5,1.558E7,
-.25282,.029534,2.073E5,1.176E7,
.12215,.034677,0.0,
VH(1,1,4)=
.2748,.1075,0.0,
.8258,.0707,-1.444E6,-6.956E7,
.8584,.0174,-1.892E6,-3.877E7,
0,-.0017,-1.989E6,1.608E4,
-.8584,.0174,-1.723E6,3.846E7,
-.8258,.0707,-1.444E6,6.956E7,
-.2748,.1075,0.0,

```

```

VH(1,1,5)=
-.10361,-.01856,0.0,
.0959,-.00459,1.241E6,2.250E7,
.6966,.00594,6.883E6,-2.488E5,
1.0,-2.426E5,-9.743E6,
.6966,-.00594,-1.026E6,1.768E4,
.0959,.00459,-1.241E6,2.250E7,
-.10361,.01856,0.0,
$
SLIST2
TITLE='CASE VERTICAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1',
ISUB=7,
PTS(1,1)=
1.0,0.0,
2,-10,0.0,
37,-50,0.0,
3,-90,0.0,
33,-100,0.0,
XMODES(1,1)=
125.1,100,0.1,
159.64,0.1,
30637,2.38E6,15.0,
VH(1,1,1)=
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
VH(1,1,2)=
-1,.02,0.0,
-.8,.02,0.0,
0,.02,0.0,
.8,.02,0.0,
1,.02,0.0,
VH(1,1,3)=
1,-.03122,0.0,
.6117,-.03028,-9.6716E5,1.1229E7,
-.7185,0.4,3121E5,6.8026E7,
.6117,.03028,9.6716E5,1.1229E7,
1,.03122,0.0,
$
SLIST2
TITLE='CASE HORIZONTAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1',
ISUB=8,
PTS(1,1)=
1.0,0.0,
2,-10,0.0,
37,-50,0.0,
3,-90,0.0,
33,-100,0.0,
XMODES(1,1)=
125.1,100,0.1,
159.64,0.1,
30637,2.38E6,15.0,
VH(1,1,1)=
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,
1.0,0.0,

```



```

VH(1,1,2)=
-1.,.02,0,0,
-.8.,.02,0,0,
0.,.02,0,0,
.8.,.02,0,0,
1.,.02,0,0,
VH(1,1,3)=
1.,-.03122,0,0,
.6117,-.03028,-9.6716E5,1.1229E7,
-.7185,0,4.3121E5,6.8026E7,
.6117,-.03028,9.6716E5,1.1229E7,
1.,.03122,0,0,
$
$LIST3
ITYPE=5,
ILEM=1,
JT=2.,39,
YS=5E5,
ZS=2.5E6,
IDAMP=1,
QLEEM=15,
QFREQ=60.4,
$
$LIST3
ITYPE=5,
ILEM=2,
JT=3.,40,
YS=5E5,
ZS=2.5E6,
IDAMP=1,
QLEEM=15,
QFREQ=60.4,
$
$LIST3
ITYPE=5,
ILEM=3,
JT=5.,2,
YS=1E6,
ZS=1E6,
IDAMP=1,
QLEEM=15,
QFREQ=60.4,
$
$LIST3
ITYPE=5,
ILEM=4,
JT=6.,3,
YS=1E6,
ZS=1E6,
IDAMP=1,
QLEEM=15,
QFREQ=60.4,
$
$LIST3
ITYPE=3,
ILEM=5,
JT=4.,1,
SK=1E6,
AK=10.,
DBAND=5,

```

```

CC=0.0,
$
$LIST4
DELTA=.00005,
TFINAL=.56,
IPRMUL=5600,
IPLMUL=5,
JPRMUL=1,
IROUT=1,
NSIG=3,
IROTI=1,
BEGTIM=0.,
BEGRPM=3000.,
TRHIS(1,1)=
-6.0,
BRPM=3000,
FRPM=3000,
SRPM=0,
UNBAL(1,1)=
0.4,5000,0,
UBALSS=
4,5000,0,
GYRO(1,1)=
4,184205,
36,184205,
IPL0T=0,
NPD(1,1)=
1,1,
1,3,
4,1,
4,3,
37,1,
37,3,
38,1,
38,3,
NEPD(1,1)=
1,2,1,
1,2,3,
2,3,1,
2,3,3,
3,5,1,
3,5,3,
4,6,1,
4,6,3,
$
END OF INPUT FILE

```

THE CURRENT PROGRAM LIMITS FOR THE INPUT VARIABLES ARE:  
 MAX NO. OF PHYSICAL POINTS NOT LOCATED ON THE MODAL SUBSYSTEMS= 10

SUBSYSTEM	MAX. NO. OF PHYSICAL POINTS	MAX. NO. OF MODES
1	10	15
2	10	15
3	10	5
4	10	15
5	10	15
6	10	5
7	20	30
8	20	30
9	20	6
10	20	30
11	10	30

MAX. NO. OF PHYSICAL POINTS FOR SUBSYSTEM 12 OTHER THAN THE C.G. POINT= 200  
 MAX. NO. OF PHYSICAL POINTS FOR SUBSYSTEM 13 OTHER THAN THE C.G. POINT= 200  
 NUMBER OF MODES FOR SUBSYSTEM 12 = 2  
 NUMBER OF MODES FOR SUBSYSTEM 13 = 2  
 MAX. NO. OF TYPE 1 PHYSICAL CONNECTING ELEMENTS= 5  
 MAX. NO. OF TYPE 2 PHYSICAL CONNECTING ELEMENTS= 10  
 MAX. NO. OF TYPE 3 PHYSICAL CONNECTING ELEMENTS= 10  
 MAX. NO. OF TYPE 4 PHYSICAL CONNECTING ELEMENTS= 4  
 MAX. NO. OF TYPE 5 PHYSICAL CONNECTING ELEMENTS= 25  
 MAX. NO. OF TYPE 6 PHYSICAL CONNECTING ELEMENTS= 5  
 MAX. NO. OF SPEED SEGMENTS= 10  
 MAX. NO. OF INPUT UNBALANCE LOADS= 20  
 MAX. NO. OF INPUT  $P \cos(WT)$  AND  $P \sin(WT)$  LOADS FOR A TRANSIENT ANALYSIS  
 OR  $P \cos(WI+PHI)$  LOADS FOR A STEADY STATE ANALYSIS= 30  
 MAX. NO. OF TIME-FORCE HISTORY LOADS= 30  
 MAX. NO. OF TIME-FORCE HISTORY TABLES= 10  
 MAX. NO. OF (TIME-FORCE) PAIRS IN EACH HISTORY TABLE= 10  
 MAX. NO. OF GYROSCOPIC LOAD LOCATIONS= 30  
 MAX. NO. OF (POINT,DIRECTION) PAIRS FOR WHICH COORDINATES, DISPLACEMENTS,  
 VELOCITIES, AND MODAL FORCES ARE WRITTEN TO THE PLOT FILE= 50  
 MAX. NO. OF (ELEMENT,POINT,DIRECTION) TRIOS FOR WHICH CONNECTING  
 ELEMENT FORCES ARE WRITTEN TO THE PLOT FILE= 50

INPUT DATA FOR POINTS NOT LOCATED ON THE MODAL SUBSYSTEMS:

POINT NUMBER	CODE (0=FREE) (1=FIXED)	COORDINATES (INCHES)-GLOBAL SYSTEM		
		X	Y	Z
39	1	-10.000	0.000	0.000
40	1	-90.000	0.000	0.000

NUMBER OF PHYSICAL POINTS NOT ON MODAL SUBSYSTEMS= 2

# DATA FOR MODAL SUBSYSTEM 1

ROTOR VERTICAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1

NUMBER OF SUBSYSTEM DIRECTIONS= 2

SUBSYSTEM DIRECTIONS:

Z (GLOBAL DIRECTION 1)  
THETA-Y (GLOBAL DIRECTION 2)

COORDINATES OF REFERENCE POINT RELATIVE TO GLOBAL SYSTEM (IN.)  
X= 0.000 Y= 0.000 Z= 0.000

POINT NUMBER	LOCAL COORDINATE SYSTEM			GLOBAL COORDINATE SYSTEM		
	X	Y	Z	X	Y	Z
4	0.000	0.000	0.000	0.000	0.000	0.000
5	-10.000	0.000	0.000	-10.000	0.000	0.000
34	-30.000	0.000	0.000	-30.000	0.000	0.000
35	-50.000	0.000	0.000	-50.000	0.000	0.000
36	70.000	0.000	0.000	-70.000	0.000	0.000
6	-90.000	0.000	0.000	-90.000	0.000	0.000
36	-100.000	0.000	0.000	-100.000	0.000	0.000

NUMBER OF SUBSYSTEM POINTS= 7

LOCAL MODE NUMBER	FREQUENCY RPM	POTENTIAL ENERGY	Q FACTOR	MODE TYPE 0=FLEXIBLE 1=RIGID BODY	GENERALIZED		GENERALIZED DAMPING VALUE (LB-SEC)/IN
					WEIGHT-LB	STIFFNESS LB/IN	
1	99.	1.920E+02	0.	1	1.372E+03	0.000E+00	0.000E+00
2	121.	3.000E+02	0.	1	1.449E+03	0.000E+00	0.000E+00
3	10065.	5.380E+05	0.	0	3.743E+02	1.076E+06	0.000E+00
4	16851.	9.900E+06	0.	0	2.457E+03	1.980E+07	0.000E+00
5	26425.	1.750E+06	0.	0	1.766E+02	3.500E+06	0.000E+00

NUMBER OF SUBSYSTEM MODES= 5

THE MODE SHAPES FOR THIS SUBSYSTEM ARE:

LOCAL MODE NUMBER	POINT NUMBER	MODAL DISPLACEMENTS		MODAL FORCES		
		GLOBAL DIRECTION 1	GLOBAL DIRECTION 2	GLOBAL DIRECTION 1	GLOBAL DIRECTION 2	THETA-Y
1	4	-1.00000	0.02000	0.000E+00	0.000E+00	0.000E+00
1	5	-0.80000	0.02000	0.000E+00	0.000E+00	0.000E+00
1	34	-0.40000	0.02000	0.000E+00	0.000E+00	0.000E+00
1	38	0.00000	0.02000	0.000E+00	0.000E+00	0.000E+00
1	35	0.40000	0.02000	0.000E+00	0.000E+00	0.000E+00
1	6	0.80000	0.02000	0.000E+00	0.000E+00	0.000E+00
1	36	1.00000	0.02000	0.000E+00	0.000E+00	0.000E+00
2	4	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00
2	5	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00
2	34	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00
2	38	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00
2	35	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00
2	6	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00

2							
3	36	1.00000	0.00000	0.00000	0.000E+00	0.000E+00	
3	4	0.12215	-0.03468	0.000E+00	0.000E+00	0.000E+00	
3	5	-0.25282	-0.02953	-2.073E+05	1.176E+07	1.176E+07	
3	34	-0.80295	-0.01610	-1.021E+05	1.568E+07	1.568E+07	
3	36	-1.00000	0.00000	-3.519E+04	1.711E+07	1.711E+07	
3	35	-0.80295	0.01610	1.586E+05	1.558E+07	1.558E+07	
3	6	-0.25282	0.02953	2.073E+05	1.176E+07	1.176E+07	
3	36	0.12215	0.03468	0.000E+00	0.000E+00	0.000E+00	
4	4	0.27480	0.10750	0.000E+00	0.000E+00	0.000E+00	
4	5	0.82580	0.07070	-1.444E+06	-6.956E+07	-6.956E+07	
4	34	0.85840	0.01740	-1.892E+06	-3.877E+07	-3.877E+07	
4	36	0.00000	-0.00170	-1.989E+06	1.608E+04	1.608E+04	
4	35	-0.85840	0.01740	-1.723E+06	3.846E+07	3.846E+07	
4	6	-0.82580	0.07070	-1.444E+06	6.956E+07	6.956E+07	
4	36	-0.27480	0.10750	0.000E+00	0.000E+00	0.000E+00	
5	4	-0.10361	-0.01856	0.000E+00	0.000E+00	0.000E+00	
5	5	0.09590	-0.00459	1.241E+06	2.250E+07	2.250E+07	
5	34	0.69660	0.00594	6.883E+06	-2.488E+05	-2.488E+05	
5	36	1.00000	0.00000	-2.426E+05	-9.743E+06	-9.743E+06	
5	35	0.69660	-0.00594	-1.026E+06	1.768E+04	1.768E+04	
5	6	0.09590	0.00459	-1.241E+06	2.250E+07	2.250E+07	
5	36	-0.10361	0.01856	0.000E+00	0.000E+00	0.000E+00	

# DATA FOR MODAL SUBSYSTEM 2

ROTOR HORIZONTAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1

NUMBER OF SUBSYSTEM DIRECTIONS= 2

SUBSYSTEM DIRECTIONS:

Y (GLOBAL DIRECTION 3)  
THETA-Z (GLOBAL DIRECTION 4)

COORDINATES OF REFERENCE POINT RELATIVE TO GLOBAL SYSTEM (IN.)  
X= 0.000 Y= 0.000 Z= 0.000

POINT NUMBER	LOCAL COORDINATE SYSTEM			GLOBAL COORDINATE SYSTEM		
	X	Y	Z	X	Y	Z
4	0.000	0.000	0.000	0.000	0.000	0.000
5	-10.000	0.000	0.000	-10.000	0.000	0.000
34	-30.000	0.000	0.000	-30.000	0.000	0.000
38	-50.000	0.000	0.000	-50.000	0.000	0.000
35	-70.000	0.000	0.000	-70.000	0.000	0.000
6	-90.000	0.000	0.000	-90.000	0.000	0.000
36	-100.000	0.000	0.000	-100.000	0.000	0.000

NUMBER OF SUBSYSTEM POINTS= 7

LOCAL MODE NUMBER	FREQUENCY RPM	POTENTIAL ENERGY	Q FACTOR	MODE TYPE 0= FLEXIBLE 1= RIGID BODY	GENERALIZED WEIGHT-LB	GENERALIZED STIFFNESS LB/IN	GENERALIZED DAMPING VALUE (LB-SEC)/IN
1	99.	1.920E+02	0.	1	1.372E+03	0.000E+00	0.000E+00
2	121.	3.000E+02	0.	1	1.449E+03	0.000E+00	0.000E+00
3	10065.	5.380E+05	0.	0	3.743E+02	1.076E+06	0.000E+00
4	16851.	9.900E+06	0.	0	2.457E+03	1.980E+07	0.000E+00
5	26425.	1.750E+06	0.	0	1.766E+02	3.500E+06	0.000E+00

NUMBER OF SUBSYSTEM MODES= 5

THE MODE SHAPES FOR THIS SUBSYSTEM ARE:

(SIGNS IN THETA-Z DIRECTION CHANGED TO OBTAIN RIGHT HAND COORDINATE SYSTEM)

LOCAL MODE NUMBER	POINT NUMBER	MODAL DISPLACEMENTS		MODAL FORCES	
		GLOBAL DIRECTION 3 Y	THETA-Z 4 V	GLOBAL DIRECTION 3 Y	THETA-Z 4 V
1	4	-1.00000	-0.02000	0.000E+00	0.000E+00
1	5	-0.80000	-0.02000	0.000E+00	0.000E+00
1	34	-0.40000	-0.02000	0.000E+00	0.000E+00
1	38	0.00000	-0.02000	0.000E+00	0.000E+00
1	35	0.40000	-0.02000	0.000E+00	0.000E+00
1	6	0.80000	-0.02000	0.000E+00	0.000E+00
1	36	1.00000	-0.02000	0.000E+00	0.000E+00
2	4	1.00000	0.00000	0.000E+00	0.000E+00
2	5	1.00000	0.00000	0.000E+00	0.000E+00
2	34	1.00000	0.00000	0.000E+00	0.000E+00
2	38	1.00000	0.00000	0.000E+00	0.000E+00
2	35	1.00000	0.00000	0.000E+00	0.000E+00

2	2	1.00000	0.00000	0.00000	0.000E+00
2	36	1.00000	0.00000	0.000E+00	0.000E+00
3	4	0.12215	0.03468	0.000E+00	0.000E+00
3	5	-0.25282	0.02953	-2.073E+05	-1.176E+07
3	34	-0.80295	0.01610	-1.021E+05	-1.568E+07
3	38	-1.00000	0.00000	3.519E+04	-1.711E+07
3	35	-0.80295	-0.01610	1.586E+05	-1.558E+07
3	6	-0.25282	-0.02953	2.073E+05	-1.176E+07
3	36	0.12215	-0.03468	0.000E+00	0.000E+00
4	4	0.27480	-0.10750	0.000E+00	0.000E+00
4	5	0.82580	-0.07070	-1.444E+06	6.956E+07
4	34	0.85840	-0.01740	-1.892E+06	3.877E+07
4	38	0.00000	0.00170	-1.989E+06	-1.608E+04
4	35	-0.85840	-0.01740	-1.723E+06	-3.846E+07
4	6	-0.82580	-0.07070	-1.444E+06	-6.956E+07
4	36	-0.27480	-0.10750	0.000E+00	0.000E+00
5	4	-0.10361	0.01856	0.000E+00	0.000E+00
5	5	0.09590	0.00459	1.241E+06	-2.250E+07
5	34	0.69660	-0.00594	6.883E+06	2.488E+05
5	38	1.00000	0.00000	-2.426E+05	9.743E+06
5	35	0.69660	0.00594	-1.026E+06	-1.768E+04
5	6	0.09590	-0.00459	-1.241E+06	-2.250E+07
5	36	-0.10361	-0.01856	0.000E+00	0.000E+00



# DATA FOR MODAL SUBSYSTEM 7

CASE VERTICAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1

NUMBER OF SUBSYSTEM DIRECTIONS= 2

SUBSYSTEM DIRECTIONS:

Z (GLOBAL DIRECTION 1)  
THETA-Y (GLOBAL DIRECTION 2)

COORDINATES OF REFERENCE POINT RELATIVE TO GLOBAL SYSTEM (IN.)  
X= 0.000 Y= 0.000 Z= 0.000

COORDINATES OF POINTS ON SUBSYSTEM (INCHES)  
GLOBAL COORDINATE SYSTEM

POINT NUMBER	LOCAL X	LOCAL Y	LOCAL Z	GLOBAL X	GLOBAL Y	GLOBAL Z
1	0.000	0.000	0.000	0.000	0.000	0.000
2	-10.000	0.000	0.000	-10.000	0.000	0.000
37	-50.000	0.000	0.000	-50.000	0.000	0.000
3	-90.000	0.000	0.000	-90.000	0.000	0.000
33	-100.000	0.000	0.000	-100.000	0.000	0.000

NUMBER OF SUBSYSTEM POINTS= 5

LOCAL MODE NUMBER	FREQUENCY RPM	POTENTIAL ENERGY	Q FACTOR	MODE TYPE 0=FLEXIBLE 1=RIGID BODY	GENERALIZED WEIGHT-LB	GENERALIZED STIFFNESS LB/IN	GENERALIZED DAMPING VALUE (LB-SEC)/IN
1	125.	1.000E+02	0.	1	4.503E+02	0.000E+00	0.000E+00
3	159.	6.400E+01	0.	1	1.784E+02	0.000E+00	0.000E+00
3	30637.	2.380E+06	15.	0	1.787E+02	4.760E+06	9.891E+01

NUMBER OF SUBSYSTEM MODES= 3

THE MODE SHAPES FOR THIS SUBSYSTEM ARE:

LOCAL MODE NUMBER	POINT NUMBER	MODAL DISPLACEMENTS		MODAL FORCES	
		GLOBAL DIRECTION 1 Z	GLOBAL DIRECTION 2 THETA-Y	GLOBAL DIRECTION 1 Z	GLOBAL DIRECTION 2 THETA-Y
1	1	1.00000	0.00000	0.000E+00	0.000E+00
1	2	1.00000	0.00000	0.000E+00	0.000E+00
1	37	1.00000	0.00000	0.000E+00	0.000E+00
1	3	1.00000	0.00000	0.000E+00	0.000E+00
1	33	1.00000	0.00000	0.000E+00	0.000E+00
2	1	-1.00000	0.02000	0.000E+00	0.000E+00
2	2	-0.80000	0.02000	0.000E+00	0.000E+00
2	37	0.00000	0.02000	0.000E+00	0.000E+00
2	3	0.80000	0.02000	0.000E+00	0.000E+00
2	33	1.00000	0.02000	0.000E+00	0.000E+00
3	1	1.00000	-0.03122	0.000E+00	0.000E+00
3	2	0.61170	-0.03028	-9.672E+05	1.123E+07
3	37	-0.71850	0.00000	4.312E+05	6.803E+07
3	3	0.61170	0.03028	9.672E+05	1.123E+07
3	33	1.00000	0.03122	0.000E+00	0.000E+00

# DATA FOR MODAL SUBSYSTEM 8

## CASE HORIZONTAL PLANE SUBSYSTEM FOR DEMONSTRATOR MODEL 1

NUMBER OF SUBSYSTEM DIRECTIONS= 2

### SUBSYSTEM DIRECTIONS:

Y (GLOBAL DIRECTION 3)  
THETA-Z (GLOBAL DIRECTION 4)

COORDINATES OF REFERENCE POINT RELATIVE TO GLOBAL SYSTEM (IN.)  
X= 0.000 Y= 0.000 Z= 0.000

POINT NUMBER	COORDINATES OF POINTS ON SUBSYSTEM (INCHES)			GLOBAL COORDINATE SYSTEM		
	LOCAL X	LOCAL Y	LOCAL Z	GLOBAL X	GLOBAL Y	GLOBAL Z
1	0.000	0.000	0.000	0.000	0.000	0.000
2	-10.000	0.000	0.000	-10.000	0.000	0.000
37	-50.000	0.000	0.000	-50.000	0.000	0.000
3	-90.000	0.000	0.000	-90.000	0.000	0.000
33	-100.000	0.000	0.000	-100.000	0.000	0.000

NUMBER OF SUBSYSTEM POINTS= 5

LOCAL MODE NUMBER	FREQUENCY RPM	POTENTIAL ENERGY	Q FACTOR	MODE TYPE 0=FLEXIBLE 1=RIGID BODY	GENERALIZED STIFFNESS LB/IN	GENERALIZED DAMPING VALUE (LB-SEC)/IN
1	125.	1.000E+02	0.	1	4.503E+02	0.000E+00
2	159.	6.400E+01	0.	1	1.784E+02	0.000E+00
3	30637.	2.380E+06	15.	0	1.787E+02	9.891E+01

NUMBER OF SUBSYSTEM MODES= 3

THE MODE SHAPES FOR THIS SUBSYSTEM ARE:

(SIGNS IN THETA-Z DIRECTION CHANGED TO OBTAIN RIGHT HAND COORDINATE SYSTEM)

LOCAL MODE NUMBER	POINT NUMBER	MODAL DISPLACEMENTS				MODAL FORCES			
		GLOBAL DIRECTION		GLOBAL DIRECTION		GLOBAL DIRECTION		GLOBAL DIRECTION	
		3 Y	4 THETA-Z	3 Y	4 THETA-Z	3 Y	4 THETA-Z	3 Y	4 THETA-Z
1	1	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
1	2	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
1	37	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
1	33	1.00000	0.00000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
2	1	-1.00000	-0.02000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
2	2	-0.80000	-0.02000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
2	37	0.00000	-0.02000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
2	33	0.80000	-0.02000	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
3	1	1.00000	0.03122	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
3	2	0.61170	0.03028	-9.672E+05	-1.123E+07	-9.672E+05	-1.123E+07	-9.672E+05	-1.123E+07
3	37	-0.71850	0.00000	4.312E+05	-6.803E+07	4.312E+05	-6.803E+07	4.312E+05	-6.803E+07
3	33	0.61170	-0.03028	9.672E+05	-1.123E+07	9.672E+05	-1.123E+07	9.672E+05	-1.123E+07
3	33	1.00000	-0.03122	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

TOTAL NUMBER OF SUBSYSTEMS= 4

SUMMARY OF THE MODES OR GENERALIZED COORDINATES:

GENERALIZED COORDINATE NUMBER	SUBSYSTEM NUMBER	GENERALIZED WEIGHT	GENERALIZED STIFFNESS	GENERALIZED DAMPING VALUE
1	1	1.372E+03	0.000E+00	0.000E+00
2	1	1.449E+03	0.000E+00	0.000E+00
3	1	3.743E+02	1.076E+06	0.000E+00
4	1	2.457E+03	1.980E+07	0.000E+00
5	1	1.766E+02	3.500E+06	0.000E+00
6	2	1.372E+03	0.000E+00	0.000E+00
7	2	1.449E+03	0.000E+00	0.000E+00
8	2	3.743E+02	1.076E+06	0.000E+00
9	2	2.457E+03	1.980E+07	0.000E+00
10	2	1.766E+02	3.500E+06	0.000E+00
11	7	4.503E+02	0.000E+00	0.000E+00
12	7	1.787E+02	4.760E+06	0.000E+00
13	7	1.787E+02	4.760E+06	9.891E+01
14	8	4.503E+02	0.000E+00	0.000E+00
15	8	1.784E+02	0.000E+00	0.000E+00
16	8	1.787E+02	4.760E+06	9.891E+01

TOTAL NUMBER OF MODES OR GENERALIZED COORDINATES= 16

SUMMARY OF THE COORDINATES FOR THE PHYSICAL POINTS:

POINT NUMBER	COORDINATES (INCHES)-GLOBAL SYSTEM		
	X	Y	Z
1	0.000	0.000	0.000
2	-10.000	0.000	0.000
3	-90.000	0.000	0.000
4	0.000	0.000	0.000
5	-10.000	0.000	0.000
6	-90.000	0.000	0.000
33	-100.000	0.000	0.000
34	-30.000	0.000	0.000
35	-70.000	0.000	0.000
36	-100.000	0.000	0.000
37	-50.000	0.000	0.000
38	-50.000	0.000	0.000
39	-10.000	0.000	0.000
40	-90.000	0.000	0.000

PHYSICAL CONNECTING ELEMENT NUMBER 1  
ELEMENT TYPE = 5 (UNCOUPLED POINT SPRING-DAMPER ELEMENT)

NUMBER OF END POINTS= 2

POINT NUMBER AT I END= 2

POINT NUMBER AT J END= 39

NUMBER OF DIRECTIONS FOR POINT AT I END= 5

DIRECTIONS FOR POINT AT I END:

X (GLOBAL DIRECTION 5)  
Y (GLOBAL DIRECTION 3)  
Z (GLOBAL DIRECTION 1)  
THETA-Y (GLOBAL DIRECTION 2)  
THETA-Z (GLOBAL DIRECTION 4)

NUMBER OF DIRECTIONS FOR POINT AT J END= 5

DIRECTIONS FOR POINT AT J END:

X (GLOBAL DIRECTION 5)  
Y (GLOBAL DIRECTION 3)  
Z (GLOBAL DIRECTION 1)  
THETA-Y (GLOBAL DIRECTION 2)  
THETA-Z (GLOBAL DIRECTION 4)

SPRING CONSTANT IN X DIRECTION= 0.000E+00 LB/IN

SPRING CONSTANT IN Y DIRECTION= 5.000E+05 LB/IN

SPRING CONSTANT IN Z DIRECTION= 2.500E+06 LB/IN

SPRING CONSTANT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB)/RAD

SPRING CONSTANT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB)/RAD

THE DAMPING FOR THIS ELEMENT IS NON-STRUCTURAL.

DAMPING IS BASED ON Q-FACTOR AND SELECTED FREQUENCY.

Q-FACTOR= 15.0

AS PER THE INPUT QFREQ VALUE:

SELECTED FREQUENCY= 60.4 HERTZ

THE DAMPING COEFFICIENTS (BASED ON THE ABOVE Q-FACTOR AND SELECTED FREQUENCY) ARE AS FOLLOWS:

DAMPING COEFFICIENT IN X DIRECTION= 0.000E+00 (LB-SEC)/IN

DAMPING COEFFICIENT IN Y DIRECTION= 8.783E+01 (LB-SEC)/IN

DAMPING COEFFICIENT IN Z DIRECTION= 4.392E+02 (LB-SEC)/IN

DAMPING COEFFICIENT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

DAMPING COEFFICIENT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

PHYSICAL CONNECTING ELEMENT NUMBER 2

ELEMENT TYPE = 5 (UNCOUPLED POINT SPRING-DAMPER ELEMENT)

NUMBER OF END POINTS= 2

POINT NUMBER AT I END= 3

POINT NUMBER AT J END= 40

NUMBER OF DIRECTIONS FOR POINT AT I END= 5

DIRECTIONS FOR POINT AT I END:

X (GLOBAL DIRECTION 5)

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

THETA-Y (GLOBAL DIRECTION 2)

THETA-Z (GLOBAL DIRECTION 4)

NUMBER OF DIRECTIONS FOR POINT AT J END= 5

DIRECTIONS FOR POINT AT J END:

X (GLOBAL DIRECTION 5)

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

THETA-Y (GLOBAL DIRECTION 2)

THETA-Z (GLOBAL DIRECTION 4)

SPRING CONSTANT IN X DIRECTION= 0.000E+00 LB/IN

SPRING CONSTANT IN Y DIRECTION= 5.000E+05 LB/IN

SPRING CONSTANT IN Z DIRECTION= 2.500E+06 LB/IN

SPRING CONSTANT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB)/RAD

SPRING CONSTANT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB)/RAD

THE DAMPING FOR THIS ELEMENT IS NON-STRUCTURAL.

DAMPING IS BASED ON Q-FACTOR AND SELECTED FREQUENCY.

Q FACTOR= 15.0

AMP PER THE INPUT OFREQ VALUE:

SELECTED FREQUENCY= 60.4 HERTZ

THE DAMPING COEFFICIENTS (BASED ON THE ABOVE Q-FACTOR AND SELECTED FREQUENCY) ARE AS FOLLOWS:

DAMPING COEFFICIENT IN X DIRECTION= 0.000E+00 (LB-SEC)/IN

DAMPING COEFFICIENT IN Y DIRECTION= 8.783E+01 (LB-SEC)/IN

DAMPING COEFFICIENT IN Z DIRECTION= 4.392E+02 (LB-SEC)/IN

DAMPING COEFFICIENT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

DAMPING COEFFICIENT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

ELEMENT TYPE = 5 (UNCOUPLED POINT SPRING-DAMPER ELEMENT)

NUMBER OF END POINTS= 2

POINT NUMBER AT I END= 5

POINT NUMBER AT J END= 2

NUMBER OF DIRECTIONS FOR POINT AT I END= 5

DIRECTIONS FOR POINT AT I END:

X (GLOBAL DIRECTION 5)

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

THETA-Y (GLOBAL DIRECTION 2)

THETA-Z (GLOBAL DIRECTION 4)

NUMBER OF DIRECTIONS FOR POINT AT J END= 5

DIRECTIONS FOR POINT AT J END:

X (GLOBAL DIRECTION 5)

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

THETA-Y (GLOBAL DIRECTION 2)

THETA-Z (GLOBAL DIRECTION 4)

SPRING CONSTANT IN X DIRECTION= 0.000E+00 LB/IN

SPRING CONSTANT IN Y DIRECTION= 1.000E+06 LB/IN

SPRING CONSTANT IN Z DIRECTION= 1.000E+06 LB/IN

SPRING CONSTANT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB)/RAD

SPRING CONSTANT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB)/RAD

THE DAMPING FOR THIS ELEMENT IS NON-STRUCTURAL.

DAMPING IS BASED ON Q-FACTOR AND SELECTED FREQUENCY.

Q-FACTOR= 15.0

AS PER THE INPUT QFREQ VALUE:

SELECTED FREQUENCY= 60.4 HERTZ

THE DAMPING COEFFICIENTS (BASED ON THE ABOVE Q-FACTOR AND SELECTED FREQUENCY) ARE AS FOLLOWS:

DAMPING COEFFICIENT IN X DIRECTION= 0.000E+00 (LB-SEC)/IN

DAMPING COEFFICIENT IN Y DIRECTION= 1.757E+02 (LB-SEC)/IN

DAMPING COEFFICIENT IN Z DIRECTION= 1.757E+02 (LB-SEC)/IN

DAMPING COEFFICIENT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

DAMPING COEFFICIENT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

PHYSICAL CONNECTING ELEMENT NUMBER 4

ELEMENT TYPE = 5 (UNCOUPLED POINT SPRING-DAMPER ELEMENT)

NUMBER OF END POINTS= 2

POINT NUMBER AT I END= 6

POINT NUMBER AT J END= 3

NUMBER OF DIRECTIONS FOR POINT AT I END= 5

DIRECTIONS FOR POINT AT I END:

X (GLOBAL DIRECTION 5)

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

THETA-Y (GLOBAL DIRECTION 2)

THETA-Z (GLOBAL DIRECTION 4)

NUMBER OF DIRECTIONS FOR POINT AT J END= 5

DIRECTIONS FOR POINT AT J END:

X (GLOBAL DIRECTION 5)

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

THETA-Y (GLOBAL DIRECTION 2)

THETA-Z (GLOBAL DIRECTION 4)

SPRING CONSTANT IN X DIRECTION= 0.000E+00 LB/IN

SPRING CONSTANT IN Y DIRECTION= 1.000E+06 LB/IN

SPRING CONSTANT IN Z DIRECTION= 1.000E+06 LB/IN

SPRING CONSTANT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB)/RAD

SPRING CONSTANT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB)/RAD

THE DAMPING FOR THIS ELEMENT IS NON-STRUCTURAL.

DAMPING IS BASED ON Q-FACTOR AND SELECTED FREQUENCY.

Q-FACTOR= 15.0

AS PER THE INPUT QFREQ VALUE:

SELECTED FREQUENCY= 60.4 HERTZ

THE DAMPING COEFFICIENTS (BASED ON THE ABOVE Q-FACTOR AND SELECTED FREQUENCY) ARE AS FOLLOWS:

DAMPING COEFFICIENT IN X DIRECTION= 0.000E+00 (LB-SEC)/IN

DAMPING COEFFICIENT IN Y DIRECTION= 1.757E+02 (LB-SEC)/IN

DAMPING COEFFICIENT IN Z DIRECTION= 1.757E+02 (LB-SEC)/IN

DAMPING COEFFICIENT IN THETA-Y DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

DAMPING COEFFICIENT IN THETA-Z DIRECTION= 0.000E+00 (IN-LB-SEC)/RAD

PHYSICAL CONNECTING ELEMENT NUMBER 5

ELEMENT TYPE = 3 (RUB ELEMENT)

NUMBER OF END POINTS= 2

POINT NUMBER AT I END= 4

POINT NUMBER AT J END= 1

NUMBER OF DIRECTIONS FOR POINT AT I END= 2

DIRECTIONS FOR POINT AT I END:

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

NUMBER OF DIRECTIONS FOR POINT AT J END= 2

DIRECTIONS FOR POINT AT J END:

Y (GLOBAL DIRECTION 3)

Z (GLOBAL DIRECTION 1)

RADIAL DEAD BAND= 5.0 MILS

LINEAR FACTOR FOR EFFECTIVE RADIAL SPRING CONSTANT=  $1.00E+06$  LB/IN  
NONLINEAR FACTOR FOR EFFECTIVE RADIAL SPRING CONSTANT= 10. IN-2

THE EFFECTIVE RADIAL SPRING CONSTANT IS A FUNCTION OF THE RELATIVE  
DISPLACEMENT MAGNITUDE, WHICH VARIES AND IS FOUND LATER.

NO DAMPING FOR THIS ELEMENT



NUMBER OF TYPE 1 PHYSICAL CONNECTING ELEMENTS= 0  
NUMBER OF TYPE 2 PHYSICAL CONNECTING ELEMENTS= 0  
NUMBER OF TYPE 3 PHYSICAL CONNECTING ELEMENTS= 1  
NUMBER OF TYPE 4 PHYSICAL CONNECTING ELEMENTS= 0  
NUMBER OF TYPE 5 PHYSICAL CONNECTING ELEMENTS= 4  
NUMBER OF TYPE 6 PHYSICAL CONNECTING ELEMENTS= 0  
TOTAL NUMBER OF PHYSICAL CONNECTING ELEMENTS= 5

```

PRINT MULTIPLE FOR STEADY STATE ANALYSIS= 1
AS PER INPUT ROUTINE OPTION IROUT = 1, IMSL SUBROUTINE ZSCNT IS USED
TO SOLVE THE NONLINEAR EQUATIONS.

THE ITERATION CRITERIA FOR THIS ANALYSIS ARE AS FOLLOWS. ITERATION
STOPS (A ROOT IS ACCEPTED) IF EITHER OF THE FOLLOWING CRITERIA ARE
SATISFIED:

A ROOT IS ACCEPTED IF TWO SUCCESSIVE APPROXIMATIONS AGREE IN THE
FIRST 3 DIGITS.

MAXIMUM NUMBER OF ITERATIONS = 200

SINCE NO INITIAL GUESSES WERE INPUTTED FOR THE RUB ELEMENT RELATIVE
DISPLACEMENTS, THE INITIAL GUESSES FOR THE FIRST SOLUTION WILL BE
THOSE THAT WOULD RESULT ASSUMING NO RUB ELEMENT FORCES FOR ANY RUB
ELEMENT.

INDEPENDENT ROTOR NUMBER (ONE FOR WHICH SPEED-TIME HISTORY IS INPUT)= 1

STEADY STATE ANALYSIS BEGINNING RPM FOR THE INDEPENDENT ROTOR= 3000.
STEADY STATE ANALYSIS FINAL RPM FOR THE INDEPENDENT ROTOR= 3000.
STEADY STATE ANALYSIS STEP RPM= 0.

SUMMARY OF THE STEADY STATE ANALYSIS UNBALANCE LOAD INPUT:

      PHASE
      POIN. MAGNITUDE ANGLE ROTOR ROTOR
      NUMBER GM-IN DEGREES NUMBER DESCRIPTION
      4      5000.      0.0      1      INDEPENDENT

TOTAL NUMBER OF STEADY STATE ANALYSIS UNBALANCE LOADS= 1

TOTAL NUMBER OF STEADY STATE ANALYSIS P*COS(W*T+PHI) LOADS= 0

SUMMARY OF THE GYROSCOPIC LOAD INPUT:

      POLAR MOMENT
      OF INERTIA
      POIN. LB-IN**2 ROTOR ROTOR
      NUMBER NUMBER DESCRIPTION
      4      184205.      1      INDEPENDENT
      36      184205.      1      INDEPENDENT

TOTAL NUMBER OF GYRO LOAD LOCATIONS= 2

THIS RUN DOES NOT PRODUCE A PLOT FILE.

```

FORCING FREQUENCY= 3000. RPM

INDEPENDENT ROTOR SPEED= 3000. RPM

SUMMARY OF THE UNBALANCE FORCES:

GLOBAL POINT NUMBER	MAGNITUDE GM-IN	PHASE ANGLE DEGREES	FORCE AMPLITUDE (POUNDS)			
			V-DIRECTION		Z-DIRECTION	
			COS COMPONENT	SIN COMPONENT	COS COMPONENT	SIN COMPONENT
4	5000.	0.	2815.522	0.000	0.000	2815.522

A RUB IS PRESENT FOR AT LEAST ONE RUB ELEMENT.  
HENCE, ITERATION IS NEEDED.

THE INITIAL GUESSES FOR THE COMPONENTS OF THE RUB ELEMENT RELATIVE  
DISPLACEMENTS, WHICH ARE BASED ON NO RUB FORCES FOR THE CURRENT FORCED  
FREQUENCY, ARE AS FOLLOWS:

RUB ELEMENT NUMBER	VERTICAL		HORIZONTAL		VERTICAL		HORIZONTAL	
	COS INCHES	COMPONENT INCHES	COS INCHES	COMPONENT INCHES	SIN INCHES	COMPONENT INCHES	SIN INCHES	COMPONENT INCHES
5	0.0018		0.0206		0.0029		0.0098	

THE COS AND SIN COMPONENTS OF THE RUB ELEMENT RELATIVE DISPLACEMENTS  
(FOUND VIA ITERATION) ARE AS FOLLOWS:

RUB ELEMENT NUMBER	VERTICAL		HORIZONTAL		VERTICAL		HORIZONTAL	
	COS INCHES	COMPONENT INCHES	COS INCHES	COMPONENT INCHES	SIN INCHES	COMPONENT INCHES	SIN INCHES	COMPONENT INCHES
5	0.0000		0.0093		0.0033		0.0020	

THE MAGNITUDES AND PHASE ANGLES OF THE RUB ELEMENT RELATIVE DISPLACEMENTS  
(FOUND VIA ITERATION) ARE AS FOLLOWS:

RUB ELEMENT NUMBER	VERTICAL		HORIZONTAL		VERTICAL		HORIZONTAL	
	MAGNITUDE INCHES	PHASE ANGLE DEGREES	MAGNITUDE INCHES	PHASE ANGLE DEGREES	MAGNITUDE INCHES	PHASE ANGLE DEGREES	MAGNITUDE INCHES	PHASE ANGLE DEGREES
	0.0033		90.6		0.0095		12.4	

BY BACK SUBSTITUTING THE RELATIVE DISPLACEMENT COMPONENTS INTO THE  
ITERATING EQUATIONS, WE CALCULATE THE FOLLOWING VALUES FOR THE F  
FUNCTION COMPONENTS. IF THE ITERATION WAS SUCCESSFUL, THESE VALUES  
WILL BE CLOSE TO ZERO:

RUB ELEMENT NUMBER	VERTICAL		HORIZONTAL		VERTICAL		HORIZONTAL	
	COS COMPONENT	COMPONENT	COS COMPONENT	COMPONENT	SIN COMPONENT	COMPONENT	SIN COMPONENT	COMPONENT
5	6.52E-05		-5.95E-05		-3.65E-05		1.22E-04	

POINT	X	MAGNITUDE OF THE PHYSICAL DISPLACEMENT		THETA-X	THETA-Y	THETA-Z
		Y	Z			

NUMBER	INCHES	INCHES	INCHES	RADIANS	RADIANS	RADIANS
1	0.00000000	0.02961165	0.00149371	0.00000000	0.00001518	0.00047620
2	0.00000000	0.02484304	0.00134282	0.00000000	0.00001515	0.00047601
3	0.00000000	0.01290204	0.00029759	0.00000000	0.00001350	0.00046346
4	0.00000000	0.03905152	0.00479490	0.00000000	0.00004576	0.00071159
5	0.00000000	0.03222258	0.00413696	0.00000000	0.00005062	0.00068996
6	0.00000000	0.01857320	0.00104488	0.00000000	0.00003134	0.00067899
33	0.00000000	0.01748987	0.00023979	0.00000000	0.00001347	0.00046326
34	0.00000000	0.01922896	0.00270847	0.00000000	0.00005104	0.00065708
35	0.00000000	0.00603551	0.00085331	0.00000000	0.00001938	0.00065049
36	0.00000000	0.02522417	0.00140993	0.00000000	0.00004519	0.00069968
37	0.00000000	0.00596311	0.00075647	0.00000000	0.00001432	0.00046973
38	0.00000000	0.00697750	0.00148085	0.00000000	0.00003758	0.00064277
39	0.00000000	0.00000000	0.00000000	0.00000000	0.00000000	0.00000000
40	0.00000000	0.00000000	0.00000000	0.00000000	0.00000000	0.00000000

PHASE ANGLE FOR THE PHYSICAL DISPLACEMENT						
POINT NUMBER	X DEGREES	Y DEGREES	Z DEGREES	THETA-X DEGREES	THETA-Y DEGREES	THETA-Z DEGREES
1	0.0	15.2	86.6	0.0	-103.9	18.4
2	0.0	14.5	87.8	0.0	-103.8	18.4
3	0.0	-154.1	131.4	0.0	-101.0	18.4
4	0.0	14.7	88.5	0.0	-117.9	17.3
5	0.0	14.2	90.1	0.0	-109.4	17.4
6	0.0	-154.1	128.9	0.0	-144.4	19.5
33	0.0	-156.1	157.2	0.0	-101.0	18.4
34	0.0	12.2	93.1	0.0	-102.5	17.8
35	0.0	-141.0	113.7	0.0	-145.5	19.1
36	0.0	-155.7	129.9	0.0	134.0	19.6
37	0.0	2.1	97.3	0.0	-102.5	18.4
38	0.0	1.8	98.3	0.0	-106.7	18.5
39	0.0	0.0	0.0	0.0	0.0	0.0
40	0.0	0.0	0.0	0.0	0.0	0.0

MAGNITUDE OF THE PHYSICAL VELOCITY						
POINT NUMBER	X IN/SEC	Y IN/SEC	Z IN/SEC	THETA-X RAD/SEC	THETA-Y RAD/SEC	THETA-Z RAD/SEC
1	0.000000	9.302765	0.469261	0.000000	0.004769	0.149603
2	0.000000	7.804665	0.421859	0.000000	0.004761	0.149542
3	0.000000	4.053292	0.093490	0.000000	0.004241	0.145599
4	0.000000	12.268387	1.506362	0.000000	0.014377	0.223553
5	0.000000	10.123014	1.299664	0.000000	0.015902	0.216756
6	0.000000	5.834939	0.328259	0.000000	0.009846	0.213312
33	0.000000	5.494601	0.075333	0.000000	0.004233	0.145537
34	0.000000	6.040950	0.850891	0.000000	0.016035	0.206428
35	0.000000	1.896109	0.268076	0.000000	0.006087	0.204357
36	0.000000	7.924401	0.442942	0.000000	0.014197	0.219810
37	0.000000	1.873365	0.237651	0.000000	0.004500	0.147570
38	0.000000	2.192045	0.465223	0.000000	0.011806	0.201933
39	0.000000	0.000000	0.000000	0.000000	0.000000	0.000000
40	0.000000	0.000000	0.000000	0.000000	0.000000	0.000000

PHASE ANGLE FOR THE PHYSICAL VELOCITY						
POINT NUMBER	X DEGREES	Y DEGREES	Z DEGREES	THETA-X DEGREES	THETA-Y DEGREES	THETA-Z DEGREES
1	0.0	-74.8	-3.4	0.0	166.1	-71.6

POINT NUMBER	X POUNDS	Y POUNDS	Z POUNDS	THETA-X IN-LB	THETA-Y IN-LB	THETA-Z IN-LB
2	0.0	-75.5	-2.2	0.0	166.2	-71.6
3	0.0	115.9	41.4	0.0	169.0	-71.6
4	0.0	-75.3	-1.5	0.0	152.1	-72.7
5	0.0	-75.8	0.1	0.0	160.6	-72.6
6	0.0	115.9	38.9	0.0	54.4	-70.5
33	0.0	113.9	67.2	0.0	169.0	-71.6
34	0.0	-77.8	3.1	0.0	167.5	-72.2
35	0.0	129.0	23.7	0.0	124.5	-70.9
36	0.0	114.3	39.9	0.0	44.0	-70.4
37	0.0	-87.9	7.3	0.0	167.5	-71.6
38	0.0	-88.2	8.3	0.0	163.3	-71.5
39	0.0	0.0	0.0	0.0	0.0	0.0
40	0.0	0.0	0.0	0.0	0.0	0.0

MAGNITUDE OF THE MODAL FORCE  
(POINTS ON PYLON AND POINTS NOT ON MODAL SUBSYSTEMS EXCLUDED)

POINT NUMBER	X POUNDS	Y POUNDS	Z POUNDS	THETA-X IN-LB	THETA-Y IN-LB	THETA-Z IN-LB
1	0.000	0.000	0.000	0.000	0.000	0.000
2	0.000	200.474	28.728	0.000	333.546	2327.564
3	0.000	200.474	28.728	0.000	333.546	2327.564
4	0.000	0.000	0.000	0.000	0.000	0.000
5	0.000	761.180	908.003	0.000	13686.061	41586.756
6	0.000	931.913	165.903	0.000	32116.031	38673.093
33	0.000	0.000	0.000	0.000	0.000	0.000
34	0.000	938.353	1883.978	0.000	7339.689	26272.189
35	0.000	1067.607	259.390	0.000	30126.276	20224.212
36	0.000	0.000	0.000	0.000	0.000	0.000
37	0.000	89.382	12.809	0.000	2020.641	14100.533
38	0.000	1158.848	572.769	0.000	21406.096	8918.533

PHASE ANGLE FOR THE MODAL FORCE

POINT NUMBER	X DEGREES	Y DEGREES	Z DEGREES	THETA-X DEGREES	THETA-Y DEGREES	THETA-Z DEGREES
1	0.0	0.0	0.0	0.0	0.0	0.0
2	0.0	-161.2	-125.6	0.0	54.4	-161.2
3	0.0	18.8	54.4	0.0	54.4	-161.2
4	0.0	0.0	0.0	0.0	0.0	0.0
5	0.0	30.6	-74.1	0.0	-52.2	-167.3
6	0.0	7.4	9.1	0.0	110.9	23.4
33	0.0	0.0	0.0	0.0	0.0	0.0
34	0.0	48.3	-79.3	0.0	64.1	-177.4
35	0.0	10.6	-34.8	0.0	105.0	37.6
36	0.0	0.0	0.0	0.0	0.0	0.0
37	0.0	18.8	54.4	0.0	54.4	-161.2
38	0.0	16.3	-61.1	0.0	97.1	132.4

THE FOLLOWING DATA IS FOR THE TYPE 3 PHYSICAL CONNECTING ELEMENTS  
(RUB ELEMENTS) (THE MAXIMUM AND MINIMUM FORCE MAGNITUDES ARE FOR  
THE HARMONICALLY AVERAGED FORCE VALUES):

ELEMENT NUMBER	MAXIMUM RELATIVE DISPLACEMENT INCHES	MINIMUM RELATIVE DISPLACEMENT INCHES	DEAD BAND INCHES	MAXIMUM CLEARANCE INCHES	MINIMUM CLEARANCE INCHES	MAXIMUM FORCE MAGNITUDE POUNDS	MINIMUM FORCE MAGNITUDE POUNDS
5	0.0095	0.0033	0.0050	0.0017	-0.0045	3605.188	371.622

INTERMITTENT RUB FOR ELEMENT NUMBER 5. NUMERICAL INTEGRATION WAS USED TO FIND THE FORCES FOR THIS ELEMENT.

THE MAGNITUDES OF THE FORCES (OBTAINED BY HARMONIC AVERAGING) THAT THE TYPE 3 PHYSICAL CONNECTING ELEMENTS (RUB ELEMENTS) EXERT ON THE ROTOR OR CASE ARE:

ELEMENT NUMBER	END	POINT NUMBER	FORCE IN GIVEN DIRECTION		
			Y POUNDS	Z POUNDS	
5	I	4	3593.189	473.788	
5	J	1	3593.189	473.788	

THE PHASE ANGLES FOR THE FORCES (OBTAINED BY HARMONIC AVERAGING) THAT THE TYPE 3 PHYSICAL CONNECTING ELEMENTS (RUB ELEMENTS) EXERT ON THE ROTOR OR CASE ARE:

ELEMENT NUMBER	END	POINT NUMBER	PHASE ANGLE IN GIVEN DIRECTION	
			Y DEGREES	Z DEGREES
5	I	4	-166.4	-114.5
5	J	1	13.6	65.5

THE MAGNITUDES OF THE FORCES THAT THE TYPE 5 PHYSICAL CONNECTING ELEMENTS (UNCOUPLED POINT SPRING-DAMPER ELEMENTS) EXERT ON THE ENGINE COMPONENTS OR GROUND ARE:

ELEMENT NUMBER	END	POINT NUMBER	FORCE IN GIVEN DIRECTION			THETA-Y IN-LB	THETA-Z IN-LB
			X POUNDS	Y POUNDS	Z POUNDS		
1	I	2	0.000	12440.423	3362.159	0.000	0.000
1	J	39	0.000	12440.423	3362.159	0.000	0.000
2	I	3	0.000	6460.837	745.103	0.000	0.000
2	J	40	0.000	6460.837	745.103	0.000	0.000
3	I	5	0.000	7392.471	2800.044	0.000	0.000
3	J	2	0.000	7392.471	2800.044	0.000	0.000
4	I	6	0.000	5679.792	748.815	0.000	0.000
4	J	3	0.000	5679.792	748.815	0.000	0.000

THE PHASE ANGLES FOR THE FORCES THAT THE TYPE 5 PHYSICAL CONNECTING ELEMENTS (UNCOUPLED POINT SPRING-DAMPER ELEMENTS) EXERT ON THE ENGINE COMPONENTS OR GROUND ARE:

ELEMENT NUMBER	END	POINT NUMBER	PHASE ANGLE IN GIVEN DIRECTION			THETA-Y DEGREES	THETA-Z DEGREES
			X DEGREES	Y DEGREES	Z DEGREES		
1	I	2	0.0	-168.6	-95.4	0.0	0.0
1	J	39	0.0	11.4	84.6	0.0	0.0
2	I	3	0.0	22.7	-51.8	0.0	0.0
2	J	40	0.0	-157.3	128.2	0.0	0.0
3	I	5	0.0	-170.0	-91.9	0.0	0.0
3	J	2	0.0	10.0	88.1	0.0	0.0
4	I	6	0.0	22.7	-55.2	0.0	0.0
4	J	3	0.0	-157.3	124.8	0.0	0.0

THE MAGNITUDES OF THE GYROSCOPIC FORCES ACTING ON THE ROTOR(S) ARE:

POINT NUMBER	ROTOR NUMBER	POLAR MOMENT OF INERTIA LB-IN**2	Y-AXIS MOMENT IN-LB	Z-AXIS MOMENT IN-LB
4	1	184205.	33480.617	2153.160
36	1	184205.	32920.087	2126.168

THE PHASE ANGLES FOR THE GYROSCOPIC FORCES ACTING ON THE ROTOR(S) ARE:

POINT NUMBER	ROTOR NUMBER	POLAR MOMENT OF INERTIA LB-IN**2	Y-AXIS PHASE ANGLE DEGREES	Z-AXIS PHASE ANGLE DEGREES
4	1	184205.	107.3	152.1
36	1	184205.	109.6	44.0

### 5.3.2 Frequency Sweep with No Rub Element Test Cases

TETRA 2 steady state analysis frequency sweep runs were made for the demonstrator model with no rub element. Two runs were made, one including gyroscopic stiffening at the fan and turbine locations, and one without. For both runs, an unbalance of 100 gram-inches was applied at the fan location on the rotor, the beginning forcing frequency (rotor speed) was 150 rpm, the step forcing frequency 50 rpm, and the final forcing frequency 15000 rpm (a solution calculated for each of 298 forcing frequencies). The output for the 298 forcing frequencies was written to the output plot file and subsequently plotted, as shown in figures 5-10 through 5-13.

The TETRA 2 results are compared to those produced by the NOVAS computer program (GE computer program for engine vibrations analysis), which are shown in figures 5-14 through 5-17. These figures are the same ones as figures 67 through 70 of reference 1, volume 3 (the original 1981 TETRA user's manual). Figure 5-10 corresponds to figure 5-14 (fan displacement without gyroscopic stiffening effects), figure 5-11 corresponds to figure 5-15 (middle of rotor displacement without gyroscopic stiffening effects), figure 5-12 corresponds to figure 5-16 (fan displacement with gyroscopic stiffening), and figure 5-13 corresponds to figure 5-17 (middle of rotor displacement with gyroscopic stiffening). As can be seen by comparing these figures, excellent agreement between TETRA 2 and the NOVAS program was obtained.



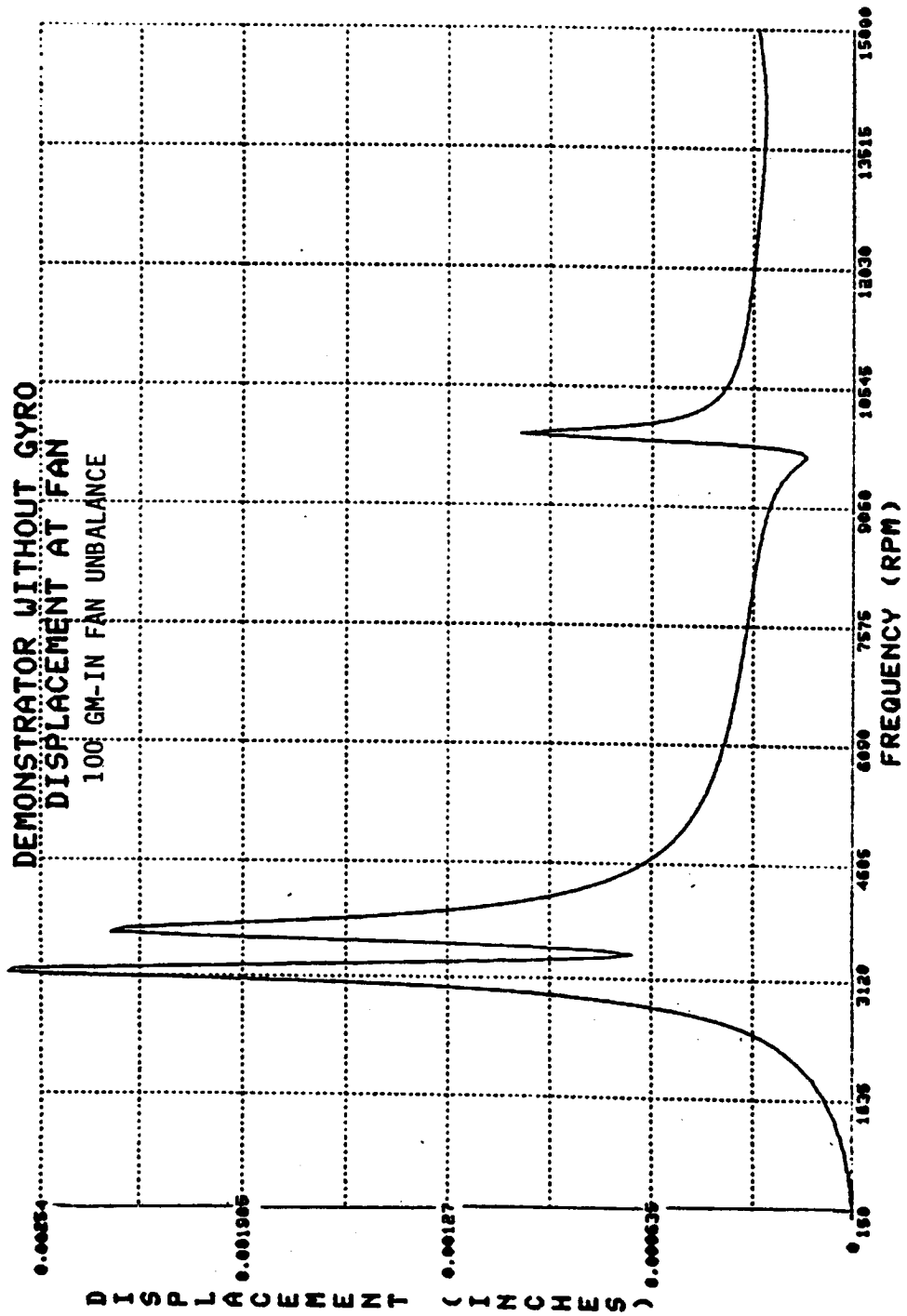


Figure 5-10. TETRA 2 Steady State Frequency Response for Demonstrator

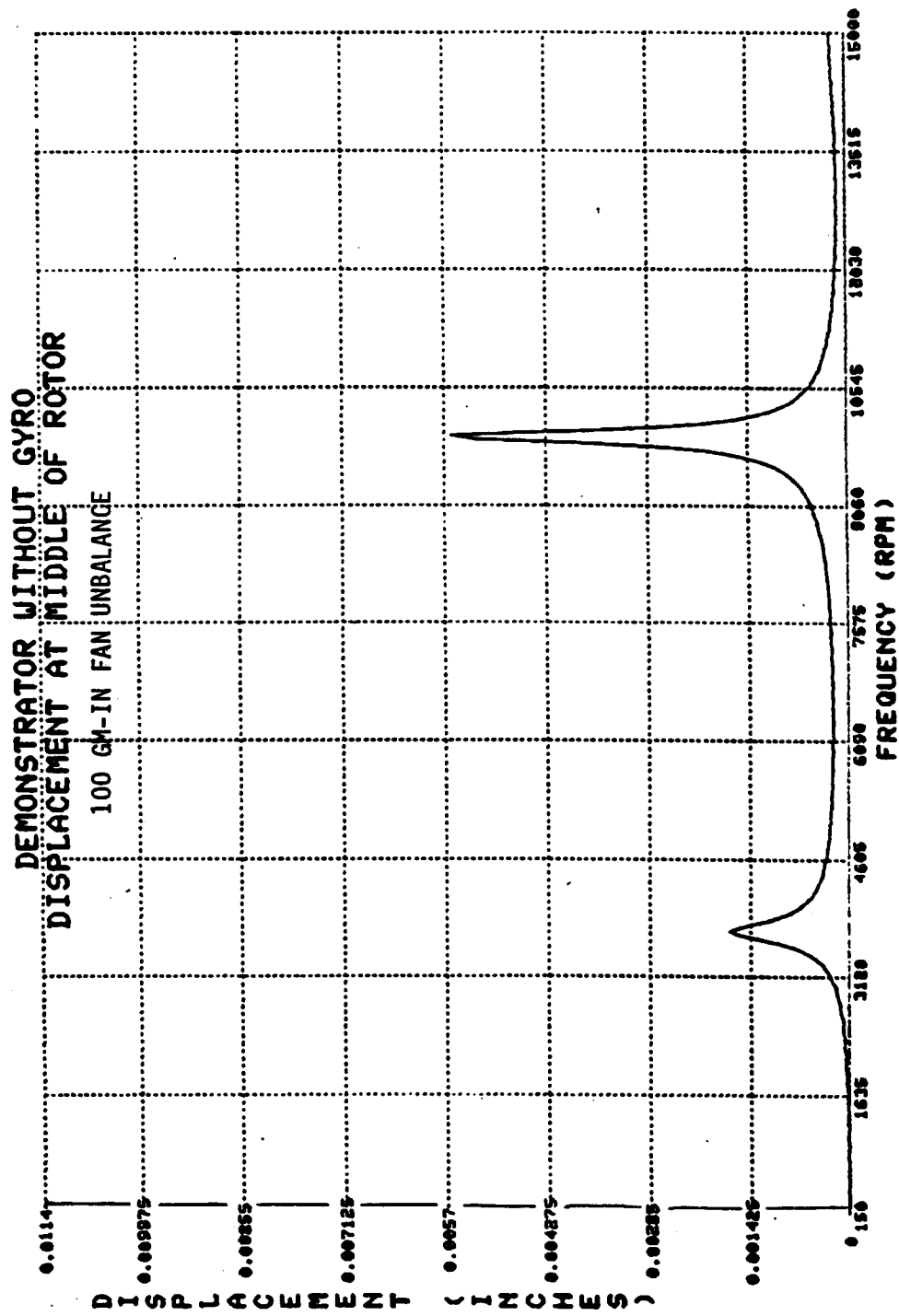


Figure 5-11. TETRA 2 Steady State Frequency Response for Demonstrator

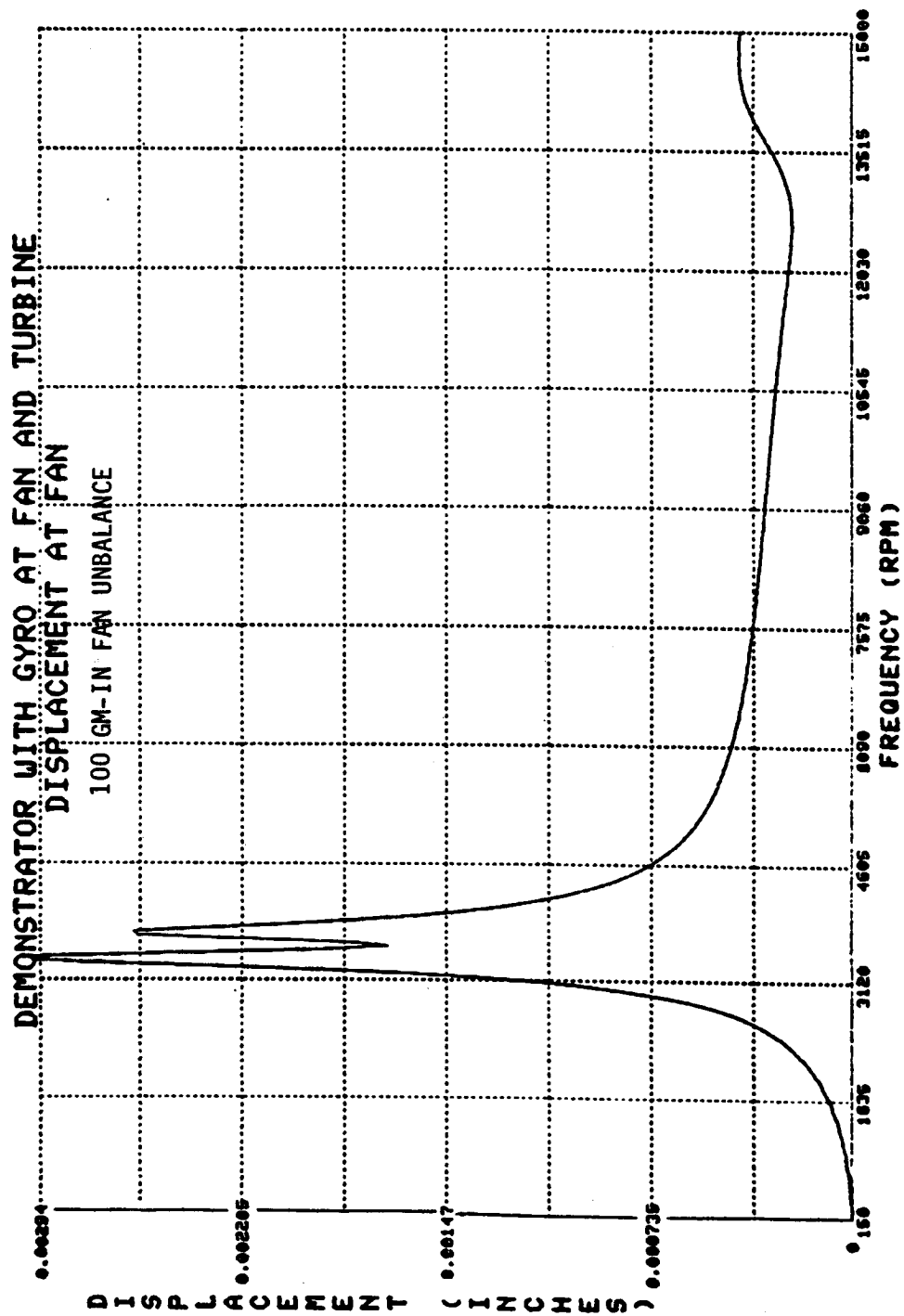


Figure 5-12. TETRA 2 Steady State Frequency Response for Demonstrator

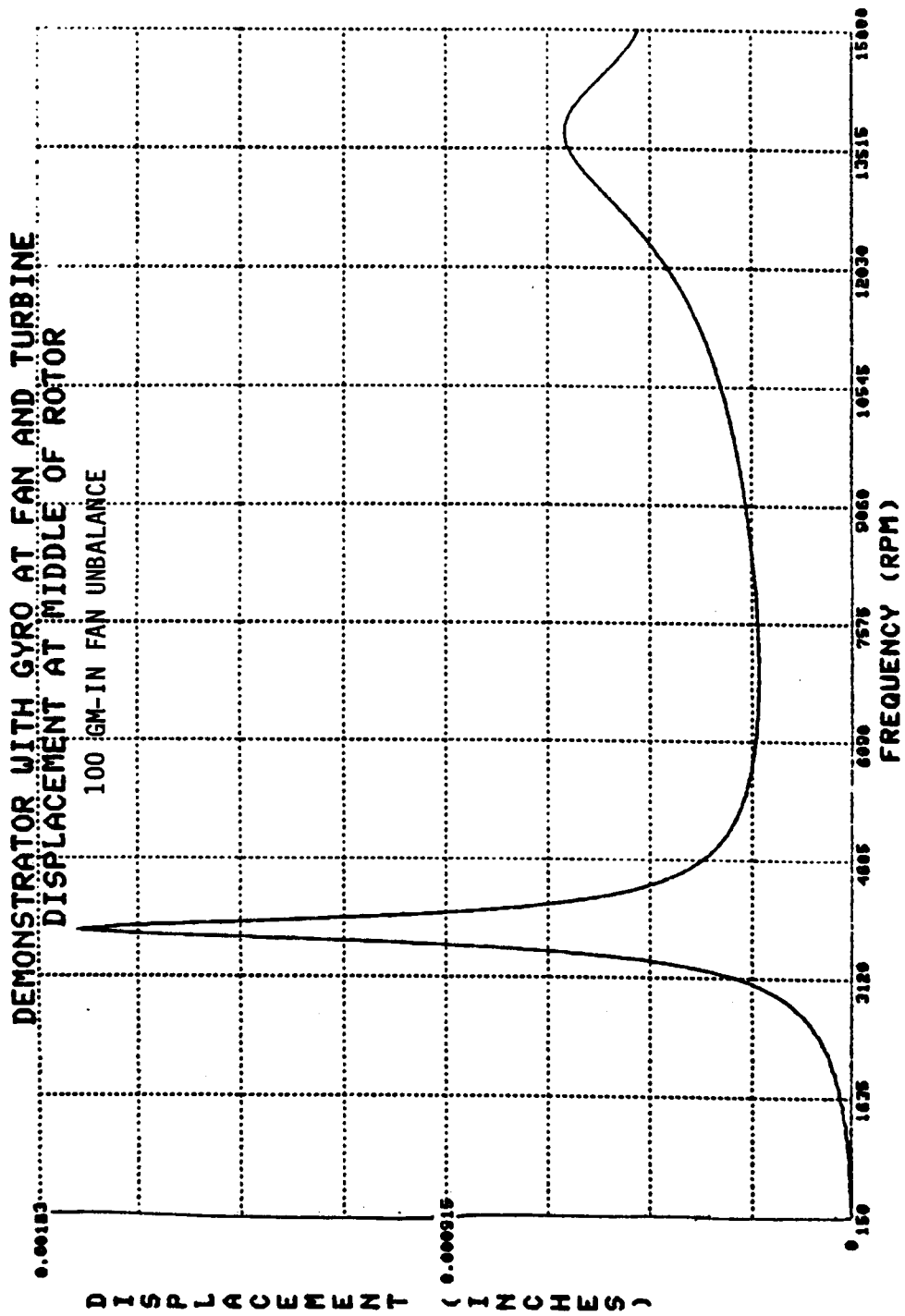


Figure 5-13. TETRA 2 Steady State Frequency Response for Demonstrator

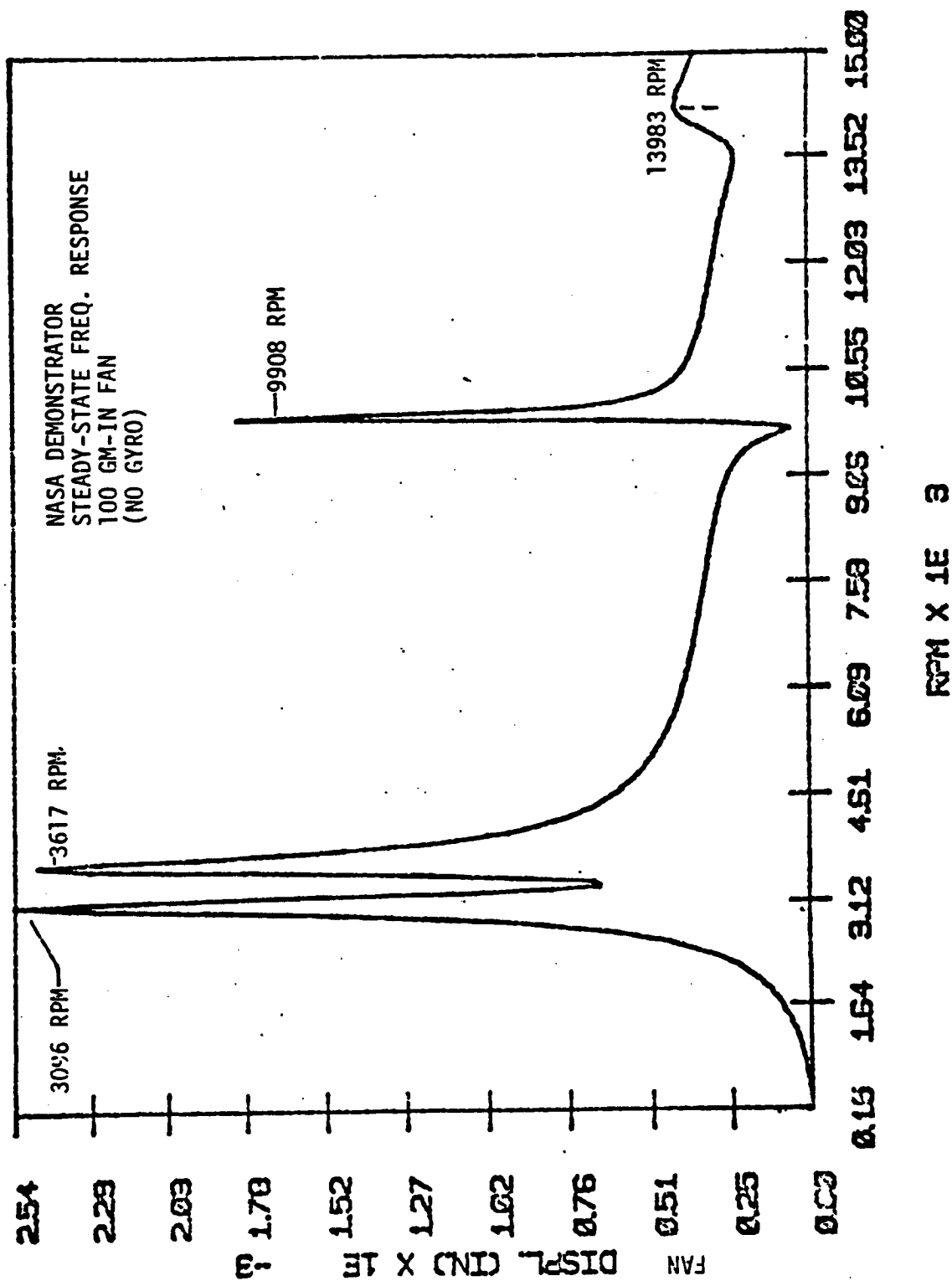


Figure 5-14. NOVAS Steady State Frequency Response for Demonstrator

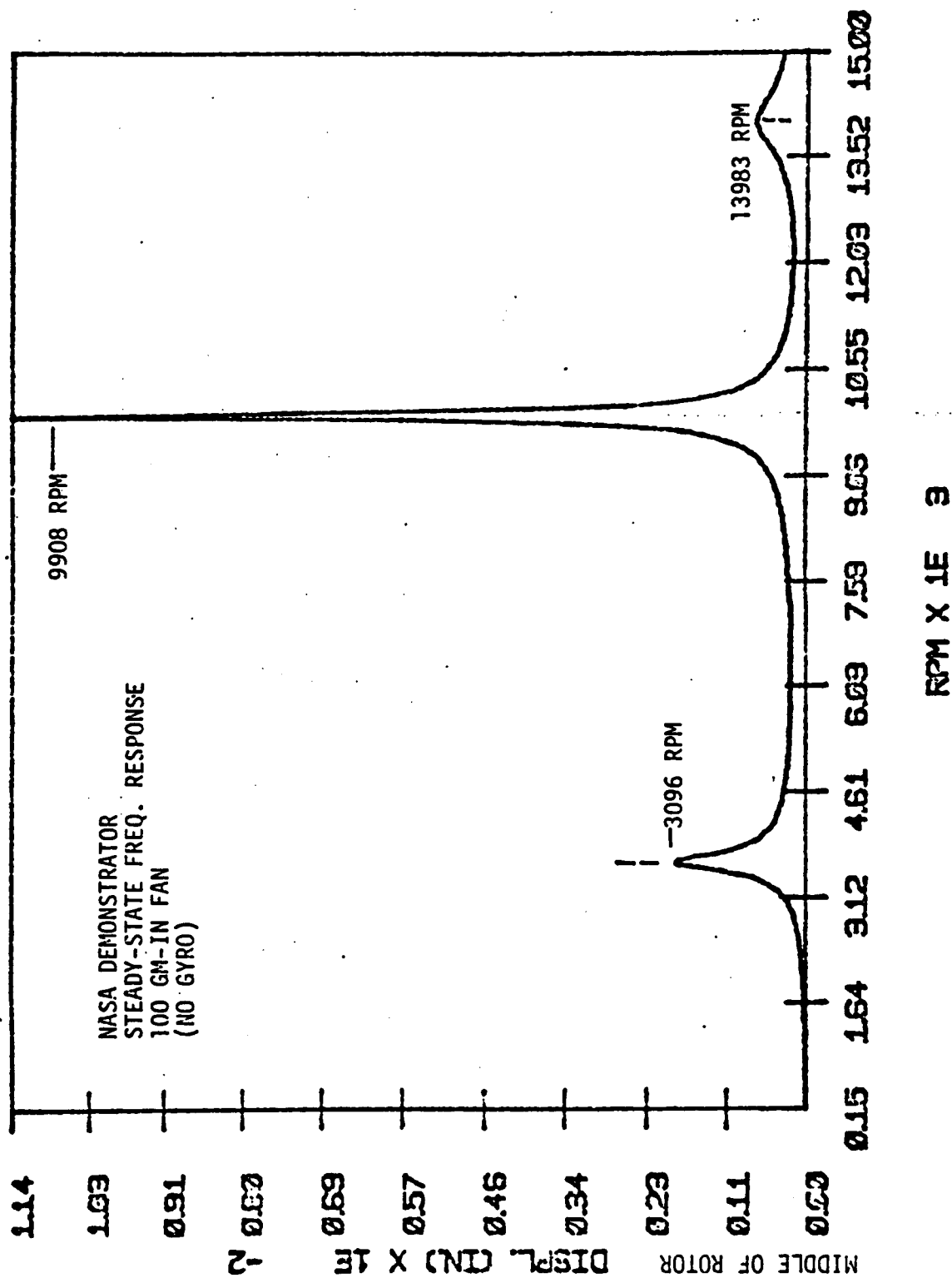


Figure 5-15. NOVAS Steady State Frequency Response for Demonstrator

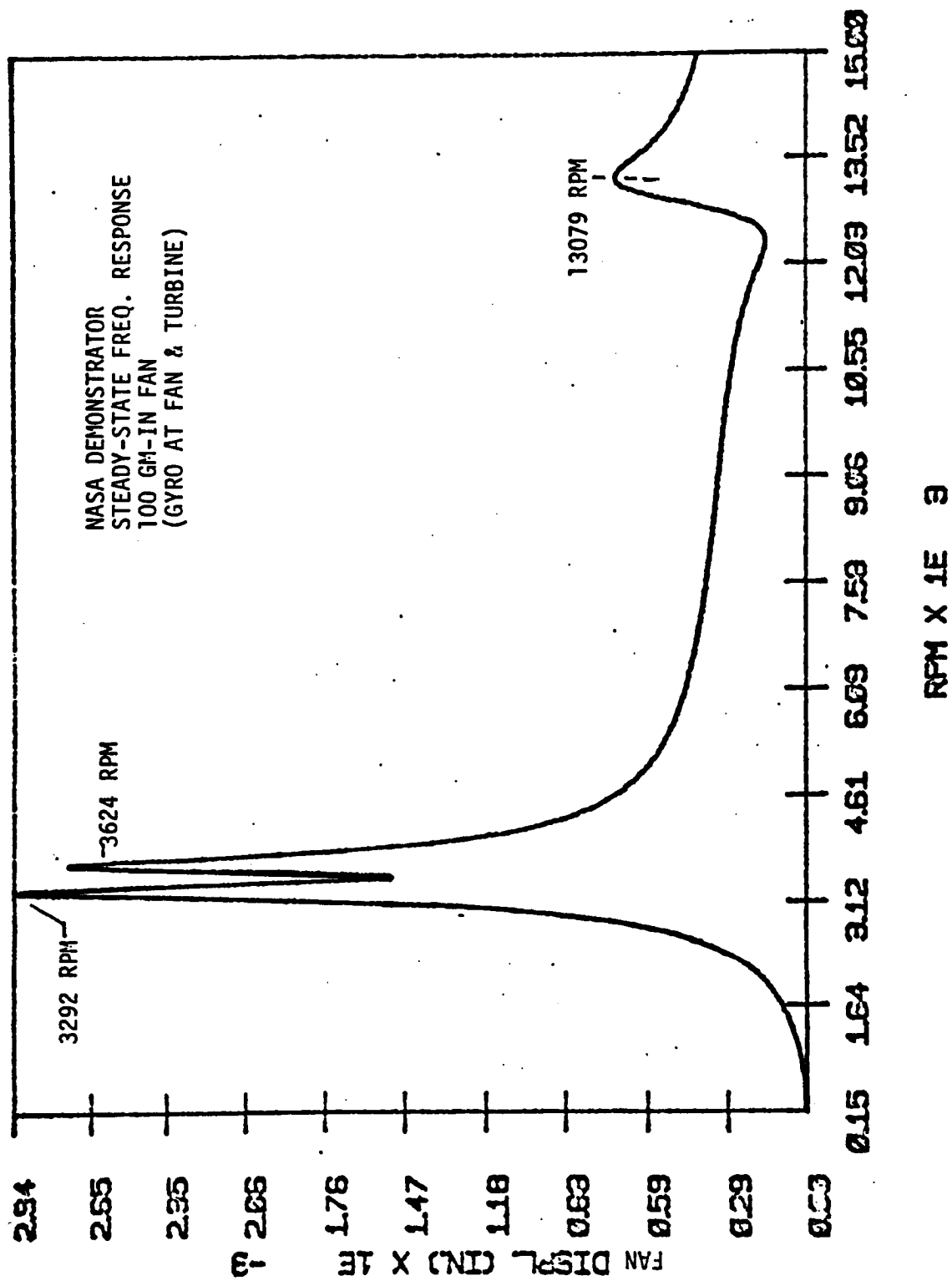


Figure 5-16. NOVAS Steady State Frequency Response for Demonstrator

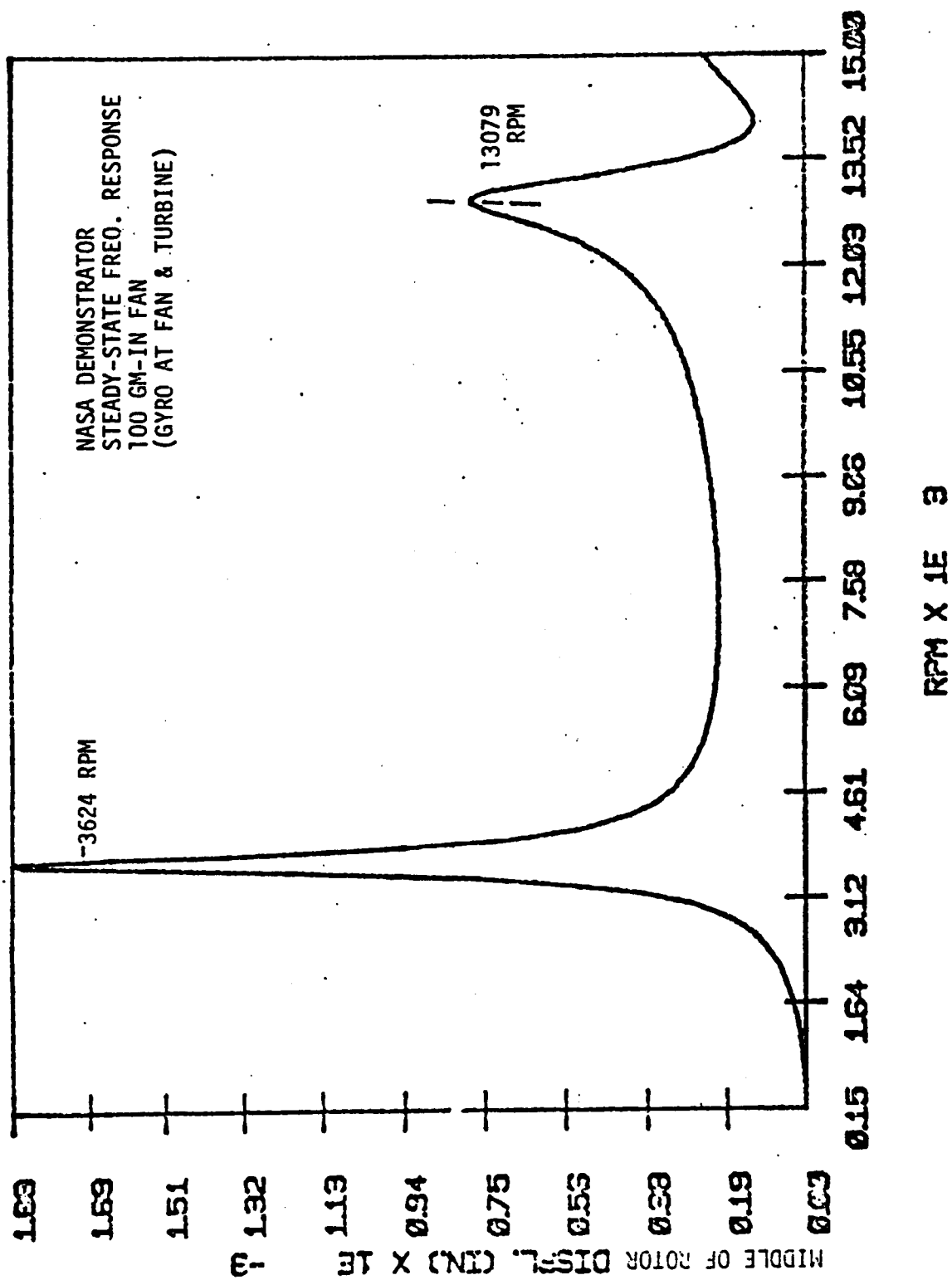


Figure 5-12. NOVAS Steady State Frequency Response for Demonstrator



### 5.3.3 Single Frequency with One Rub Element Test Cases

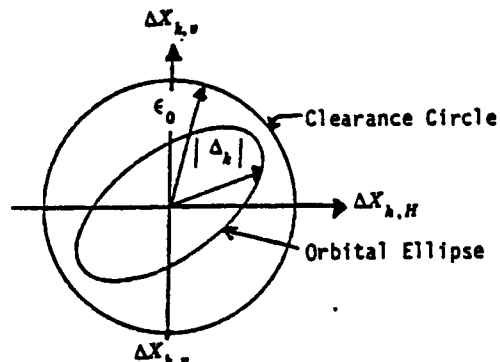
Several runs of the demonstrator model with one nonlinear spring (rub element) located at the fan were made. One of the prime reasons for these runs was to provide a comprehensive checkout of the rub element, including the dead band feature of the rub element, which was not exercised in the simpler test cases reported in the previous sections.

For each rub element, TETRA 2 recognizes four possible rub categories: no rub, continual rub with dead equal to zero, continual rub with dead band not equal to zero, and intermittent rub. For each forcing frequency, TETRA 2 compares the minimum and maximum relative displacement magnitude and the dead band of each rub element to determine which of the categories is present for each of the rub elements. These four possible rub element categories are illustrated in figure 5-18. Knowing which rub category applies is important, because a different method is used to find the rub element physical forces for each of the four rub element categories. For more details about the four rub categories and the methods used for each category, see section 5.3 of volume 1 of this report.

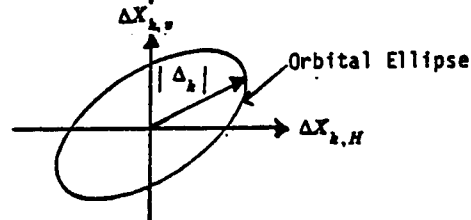
To fully check out the rub element, demonstrator model runs (one steady state run and one transient analysis run) were made for each of the four possible rub element categories. These runs were made with a rotor speed (forcing frequency) of 3000 rpm, and an unbalance of 5000 gm-in applied at the fan location on the rotor. The dead band and the spring rates were adjusted to produce a run for each of the four rub categories.

The results when a rub occurs could not be checked against another computer program, because we currently do not have any other computer program with this capability. However, what we did do is compare the results of the TETRA 2 transient analysis option when taken to steady state conditions (that is, run long enough for the transient analysis part to die out) with the direct steady state solution. Since greatly different methods of solution are utilized in the transient analysis and the steady state analysis options of TETRA 2, the fact that the two methods agree so very well is most encouraging.

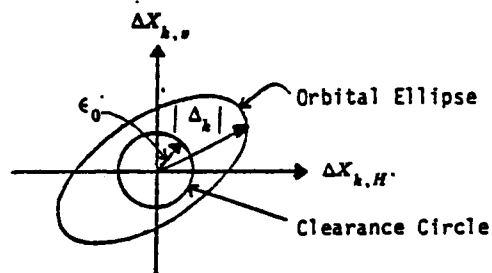
NO RUB



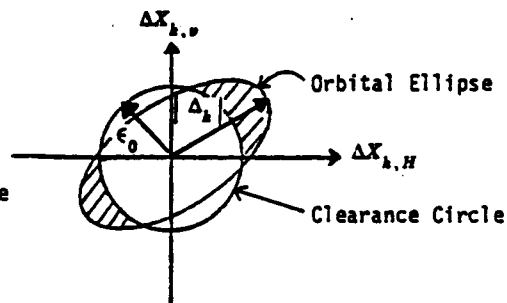
CONTINUAL RUB  
WITH  $\epsilon_0 = 0$



CONTINUAL RUB  
WITH  $\epsilon_0 > 0$



INTERMITTENT RUB  
(shaded areas indicate  
where rub occurs)



Nomenclature:

$\epsilon_0$  = Dead Band

$|\Delta_k|$  = Relative displacement magnitude for the k'th rub element.

$\Delta X_{k,H}$  = Relative displacement of the k'th rub element in the horizontal direction

$\Delta X_{k,v}$  = Relative displacement of the k'th rub element in the vertical direction

FIGURE 5-18. FOUR POSSIBLE RUB CATEGORIES

#### 5.3.3.1 No Rub

A steady state analysis test case was run where a rub element located at the fan was present but no rub occurred because the dead band which was inputted always exceeded the rub element relative displacement magnitude. Two methods were used to verify the results of this test case.

First, since no rub occurred, the rub element was never activated, and the rub element physical forces set to zero by the program. Thus, this case should produce the same results as for a TETRA run with no rub element, and remember, the demonstrator with no rub element was verified by comparing TETRA 2 results with that of the NOVAS computer program (see section 5.3.2). A TETRA 2 run with the same input except omitting the rub element confirmed that exactly the same results are obtained either way.

The second way that this test case was verified was by comparing the results of the steady state analysis run with the results of a transient analysis run taken to steady state conditions. The transient analysis run results are shown in figures 5-19 through 5-22. The transient analysis run was stopped at .5 seconds, by which time the transient effect had almost completely died out, as can be seen from these plots.

These results are summarized in table 5-1 below. The steady state analysis values in this table are exact numbers from the printout of the run. The transient analysis run numbers, however, had to be estimated by reading the peak values in the steady state region (.4 to .5 seconds) of figures 5-19 through 5-22. Although it is hard to read the figures very accurately, it is evident that the steady state and transient analysis runs agree quite well.

Table 5-1. Demonstrator Model Results with No Rub

<u>Location</u>	<u>Maximum Vertical Displacement (inches)</u>		<u>Transient Analysis Figure Number</u>
	<u>Steady State Analysis Run</u>	<u>Transient Analysis Run</u>	
Rotor at fan station	.0430	.043	5-19
Case at fan station	.0287	.028	5-20
Rotor at midpoint station	.0105	.0105	5-21
Case at midpoint station	.00869	.087	5-22

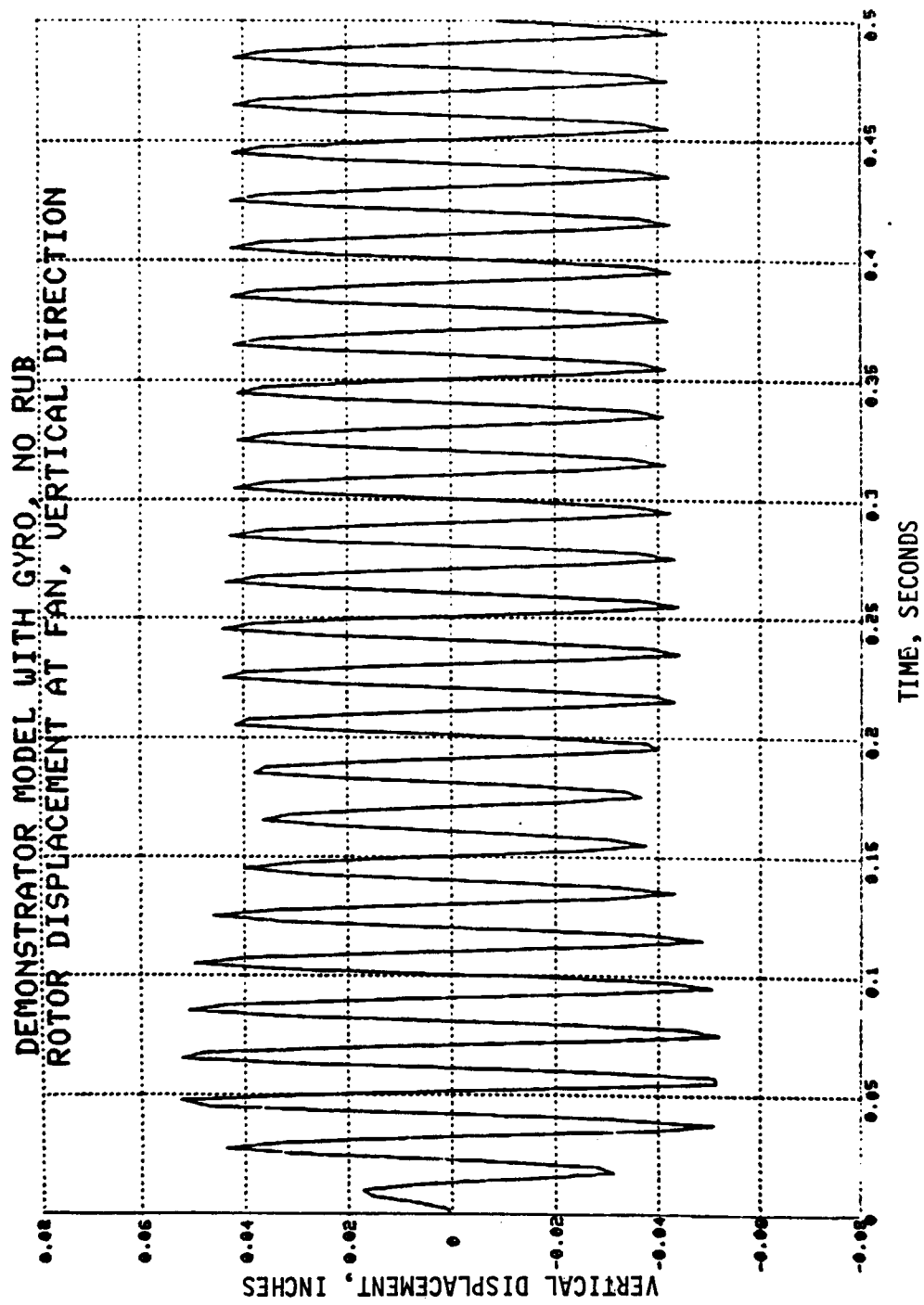


Figure 5-19. Demonstrator Transient Analysis Response Plot

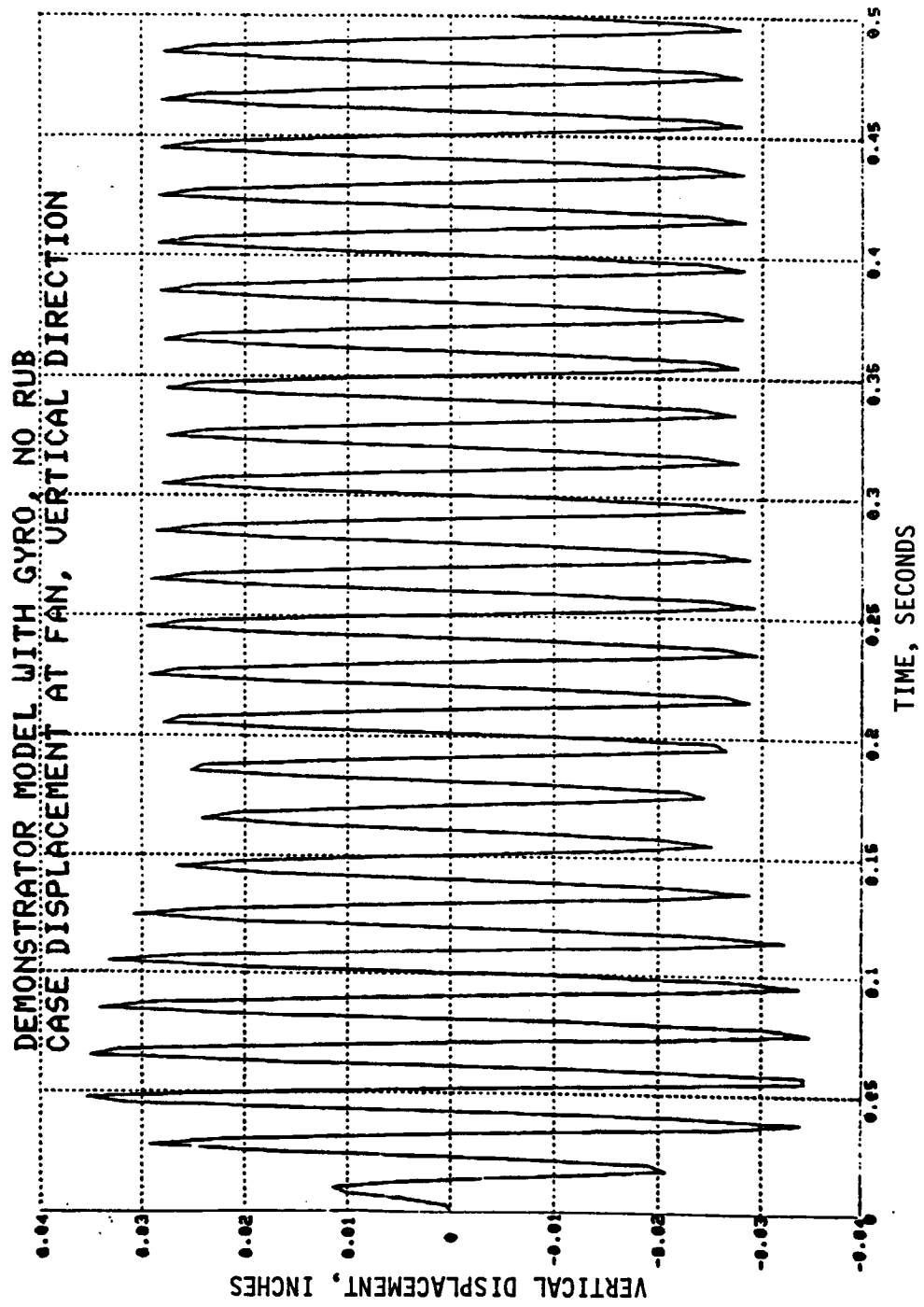


Figure 5-20. Demonstrator Transient Analysis Response Plot

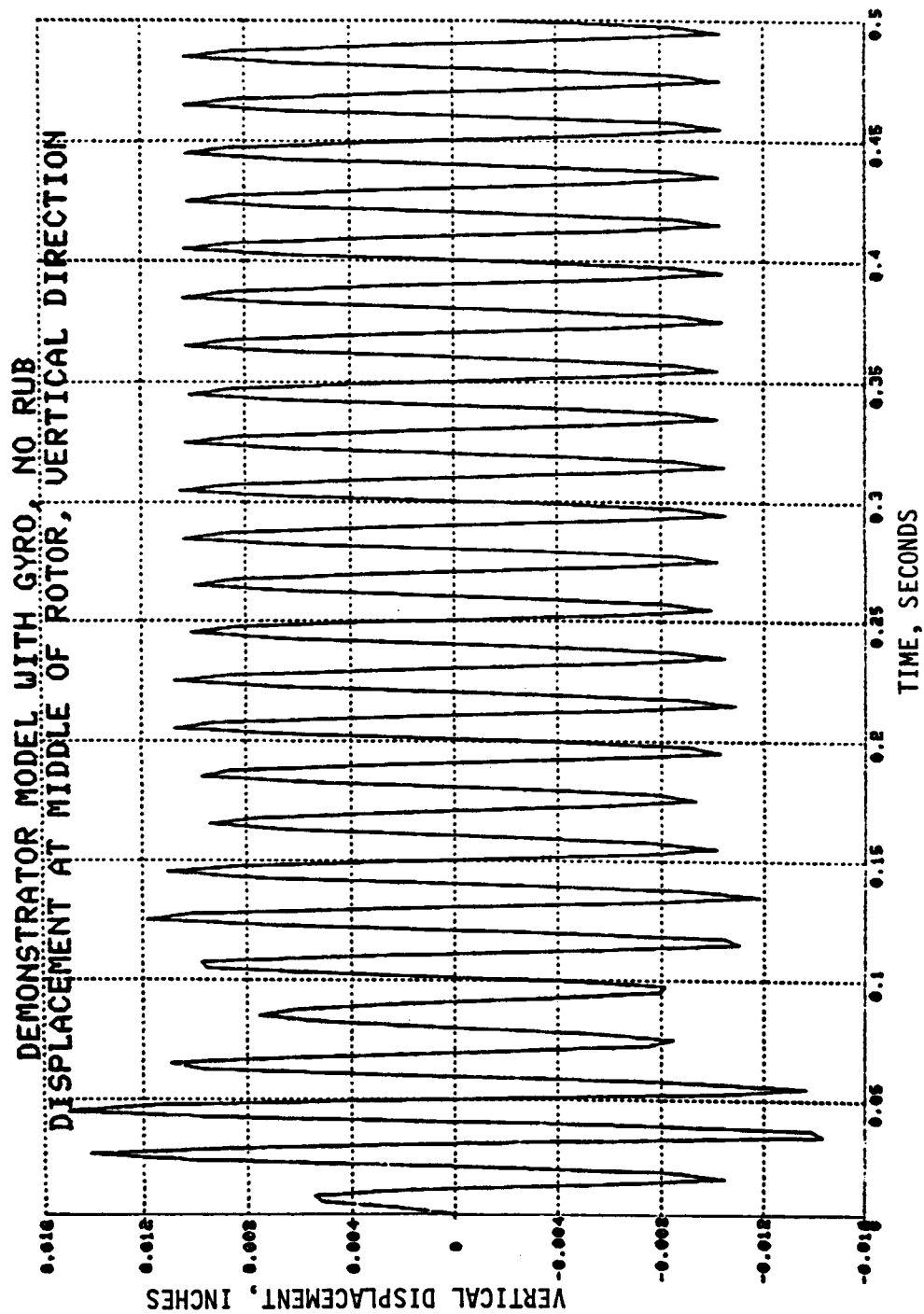


Figure 5-21. Demonstrator Transient Analysis Response Plot

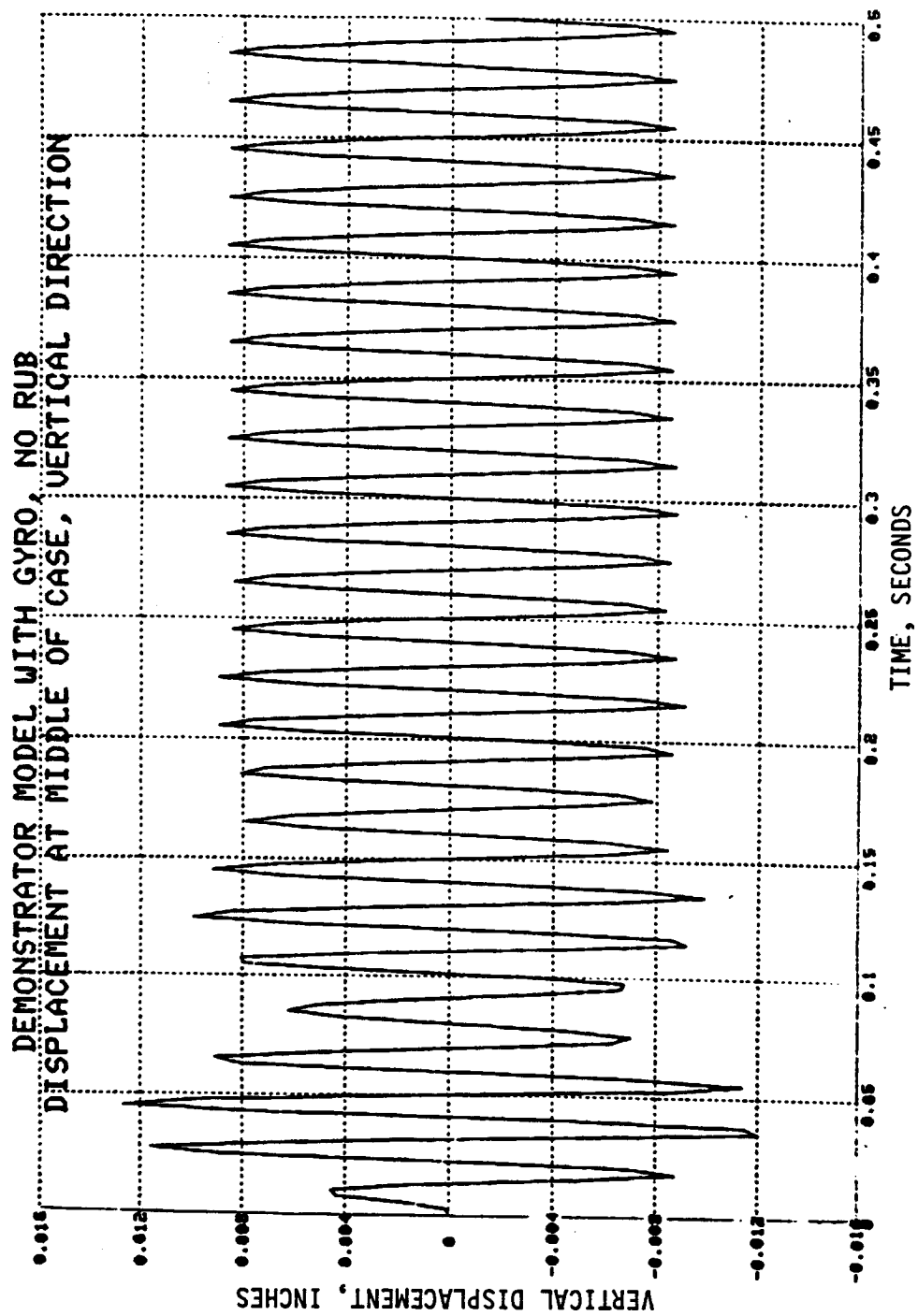


Figure 5-22. Demonstrator Transient Analysis Response Plot

### 5.3.3.2 Continual Rub with Dead Band Equal to Zero

Next, a steady state analysis test case was run for the demonstrator model where the rub element, which was located at the fan, had an input dead band equal to zero. Since the input dead band was zero, a continual rub of course resulted. A transient analysis run taken to steady state conditions was also run to verify the results. The transient analysis run results are shown in figures 5-23 through 5-26. The transient analysis run was stopped at .5 seconds, by which time the transient effect had almost completely died out, as can be seen from these plots.

The results are summarized in table 5-2 below. The steady state analysis values in this table are exact numbers from the printout of the run. The transient analysis run numbers, however, had to be estimated by reading the peak values in the steady state region (.4 to .5 seconds) of figures 5-23 through 5-26. Although it is hard to read the figures very accurately, again it is evident that the steady state and transient analysis runs agree quite well.

Table 5-2. Demonstrator Model Results for Continual Rub with Dead Band Equal to Zero

<u>Location</u>	<u>Maximum Vertical Displacement (inches)</u>		<u>Transient Analysis Figure Number</u>
	<u>Steady State Analysis Run</u>	<u>Transient Analysis Run</u>	
Rotor at fan station	.0234	.023	5-23
Case at fan station	.0195	.019	5-24
Rotor at midpoint station	.00649	.0066	5-25
Case at midpoint station	.00564	.0057	5-26



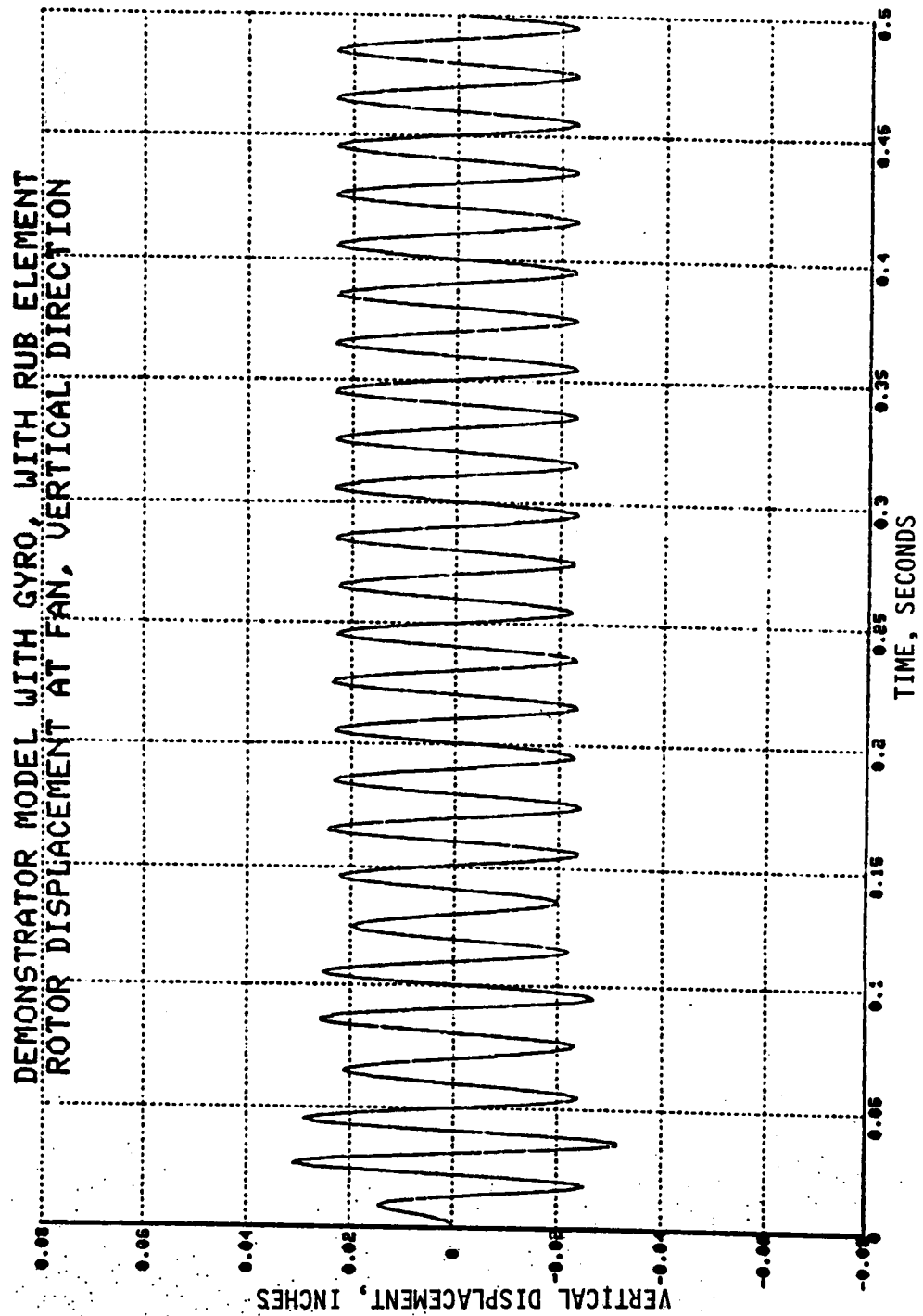


Figure 5-23. Demonstrator Transient Analysis Response Plot

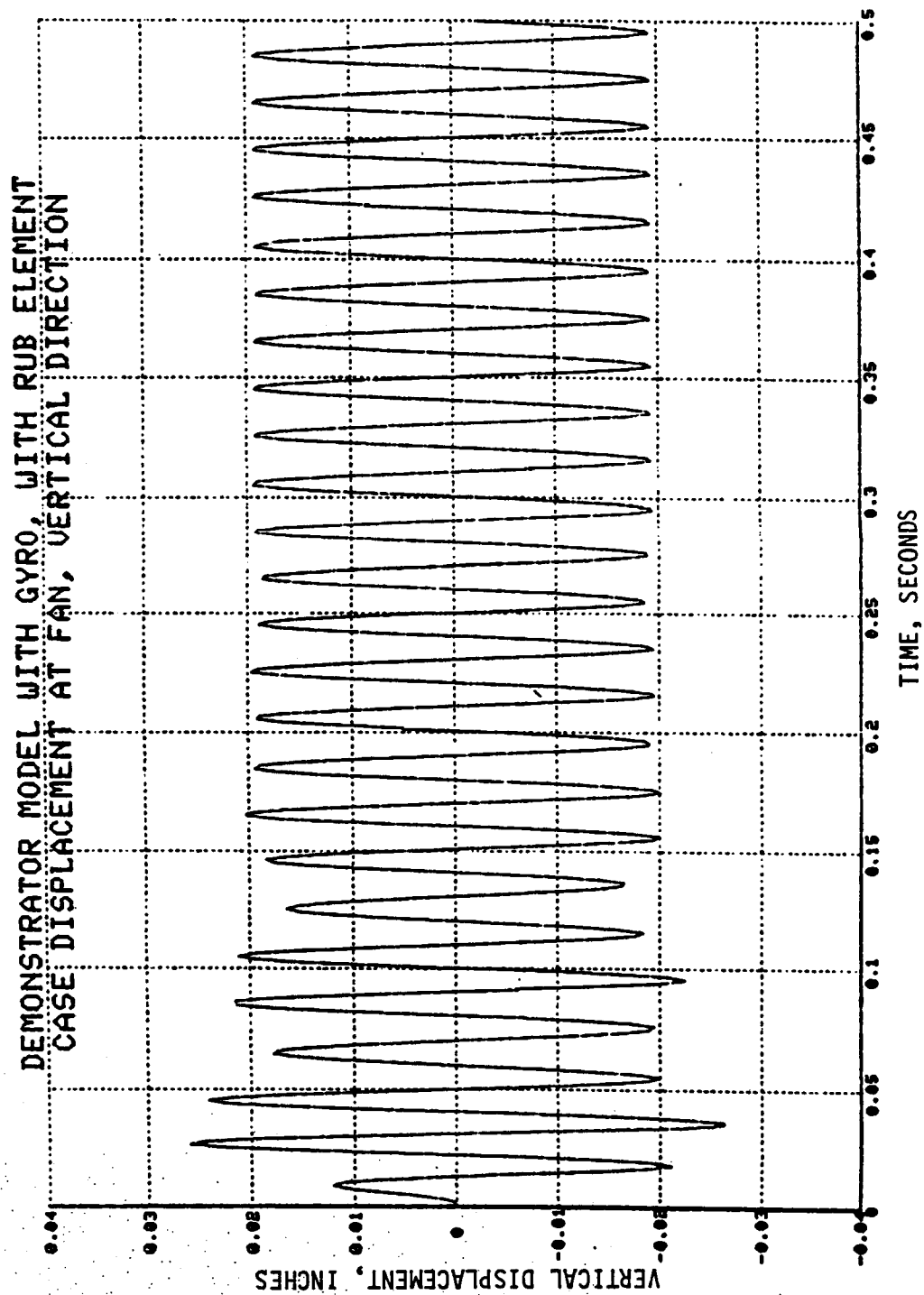


Figure 5-24. Demonstrator Transient Analysis Response Plot

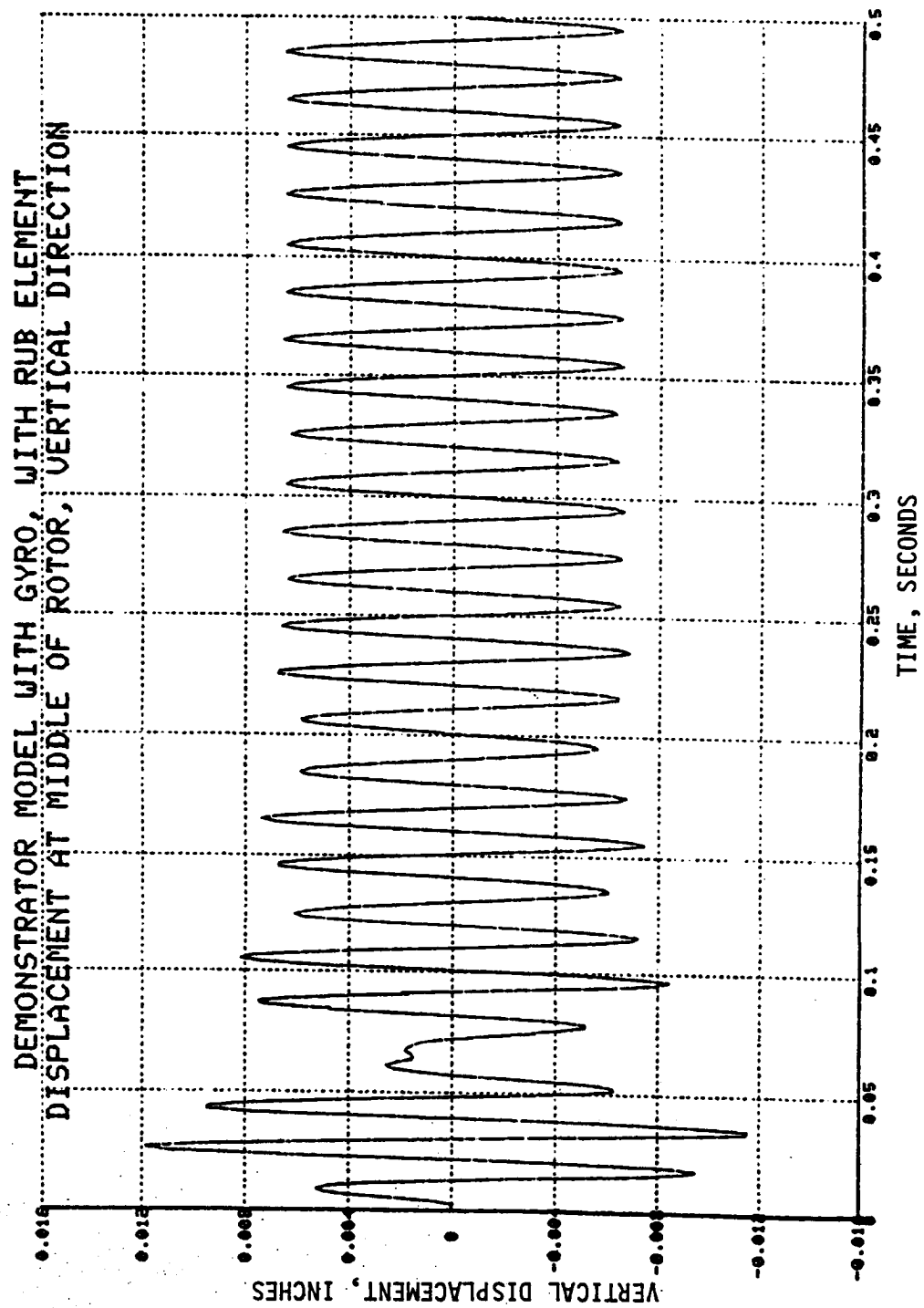


Figure 5-25. Demonstrator Transient Analysis Response Plot

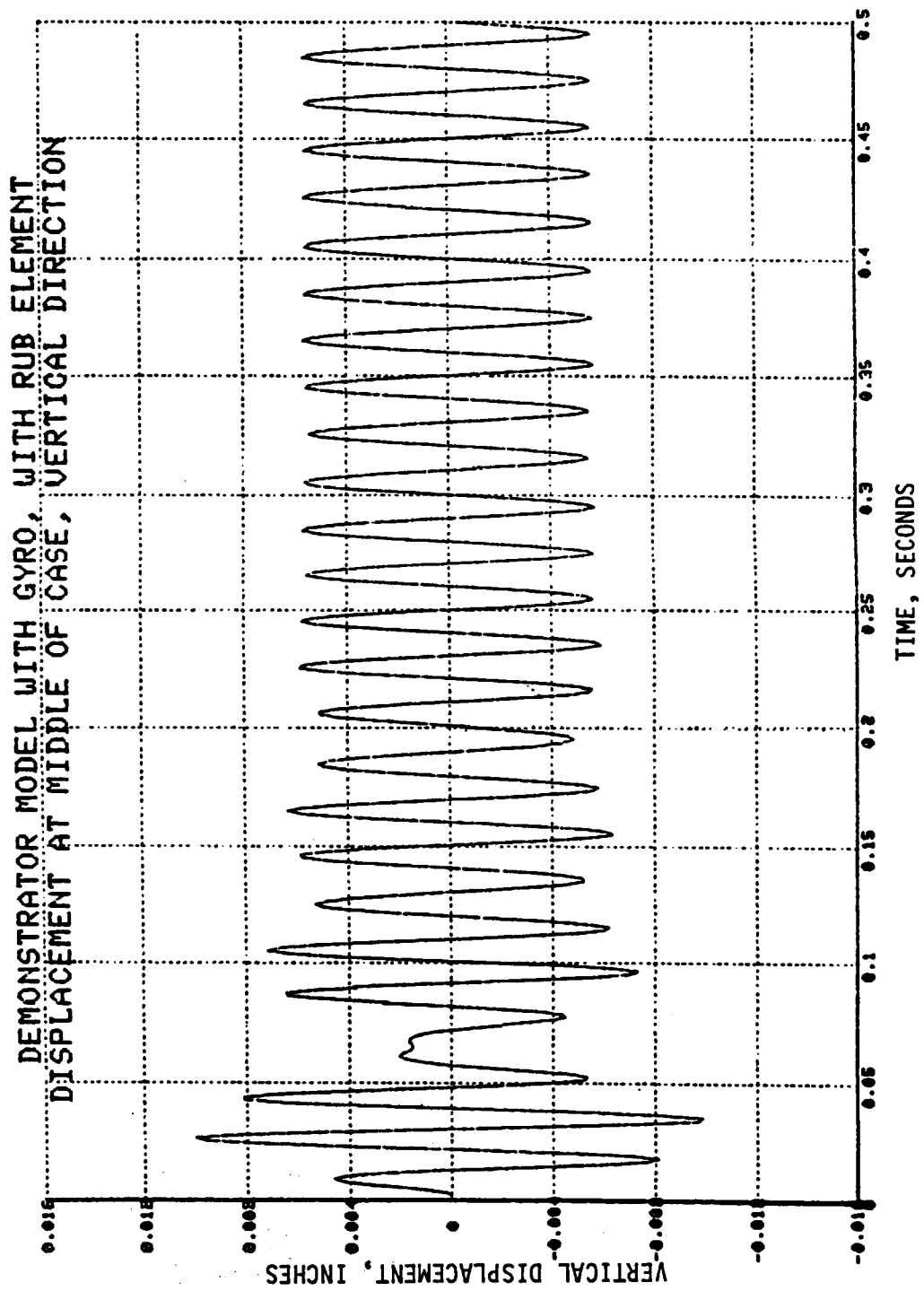


Figure 5-26. Demonstrator Transient Analysis Response Plot

### 5.3.3.3 Continual Rub with Dead Band Not Equal to Zero

Next, a steady state analysis test case was run for the demonstrator model where the rub element, which was located at the fan, had an input dead band not equal to zero. The magnitude of the input dead band compared to the minimum and maximum relative displacement magnitudes was such that a continual rub resulted for this case. A transient analysis run taken to steady state conditions was also run to verify the results. The transient analysis run results are shown in figures 5-27 through 5-30. The transient analysis run was stopped at .56 seconds, by which time the transient effect had almost completely died out, as can be seen from these plots.

The results are summarized in table 5-3 below. The steady state analysis values in this table are exact numbers from the printout of the run. The transient analysis run numbers, however, had to be estimated by reading the peak values in the steady state region (.4 to .56 seconds) of figures 5-27 through 5-30. Although it is hard to read the figures very accurately, again it is evident that the agreement between the steady state and transient analysis runs is superb.

Table 5-3. Demonstrator Model Results for Continual Rub with Dead Band Not Equal to Zero

<u>Location</u>	<u>Maximum Vertical Displacement (inches)</u>		<u>Transient</u>
	<u>Steady State</u> <u>Analysis Run</u>	<u>Transient</u> <u>Analysis Run</u>	<u>Analysis</u> <u>Figure</u> <u>Number</u>
Rotor at fan station	.0303	.030	5-27
Case at fan station	.0227	.022	5-28
Rotor at midpoint station	.00789	.0079	5-29
Case at midpoint station	.00672	.0067	5-30

Also shown are some orbit plots for this case, which are helpful in visualizing what is happening. The first orbit plot, figure 5-31, shows the vertical versus the horizontal displacement of the points located on the rotor and case at the fan station (these points are also the joints of the rub element). This plot is for the transient analysis run, and covers through .05 seconds of the run in order to show clearly the early transient portion of the run. The displacements are 0 at time = 0 (time = 0 represents the time of fan blade loss), then displacements increase as is shown by the movement out from the center of the plot. As can be seen from the plot, the rotor and case points move almost parallel to one another, with the rotor displacement larger than the case displacement.

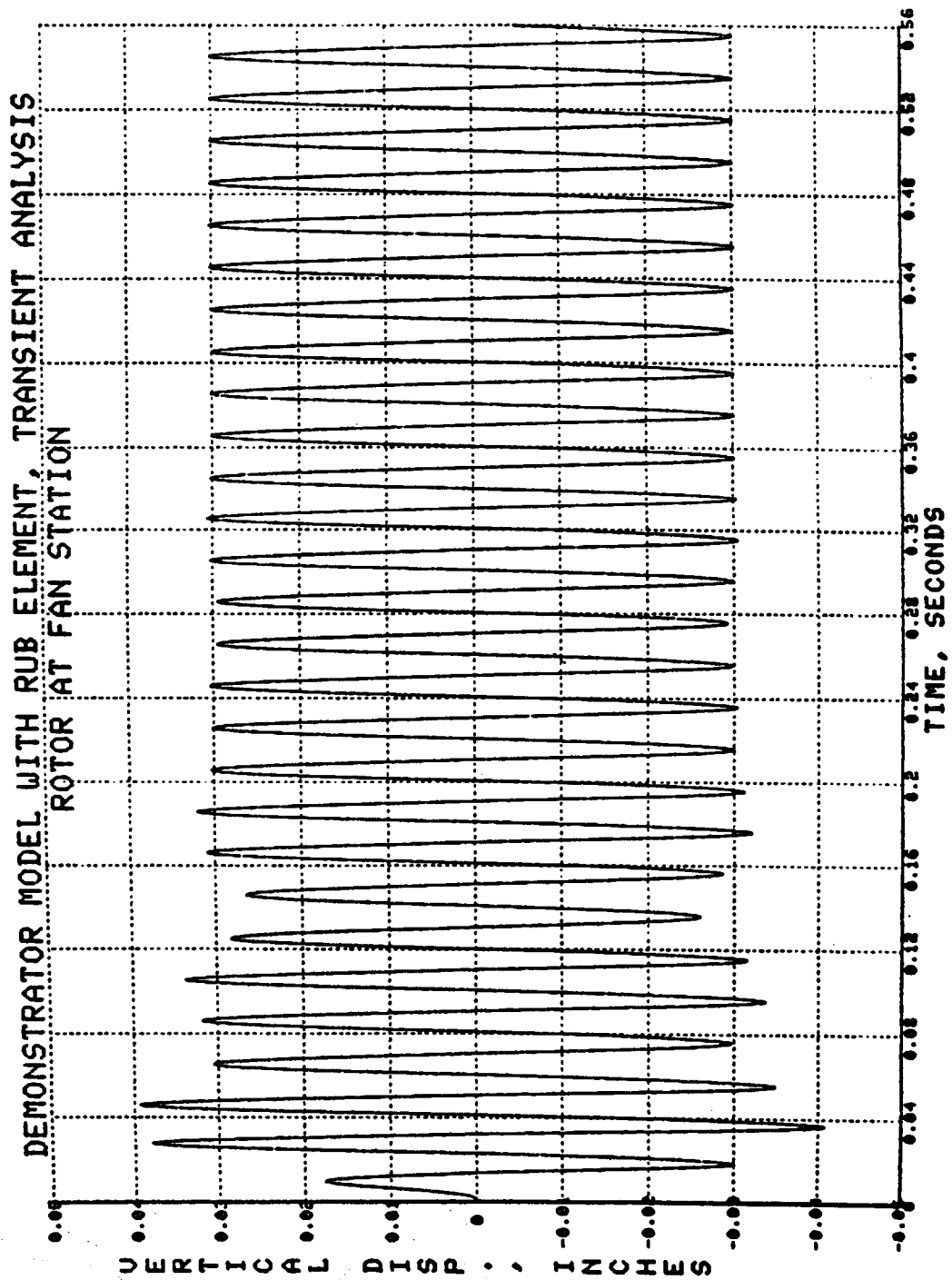


FIGURE 5-27. DEMONSTRATOR TRANSIENT ANALYSIS RESPONSE PLOT

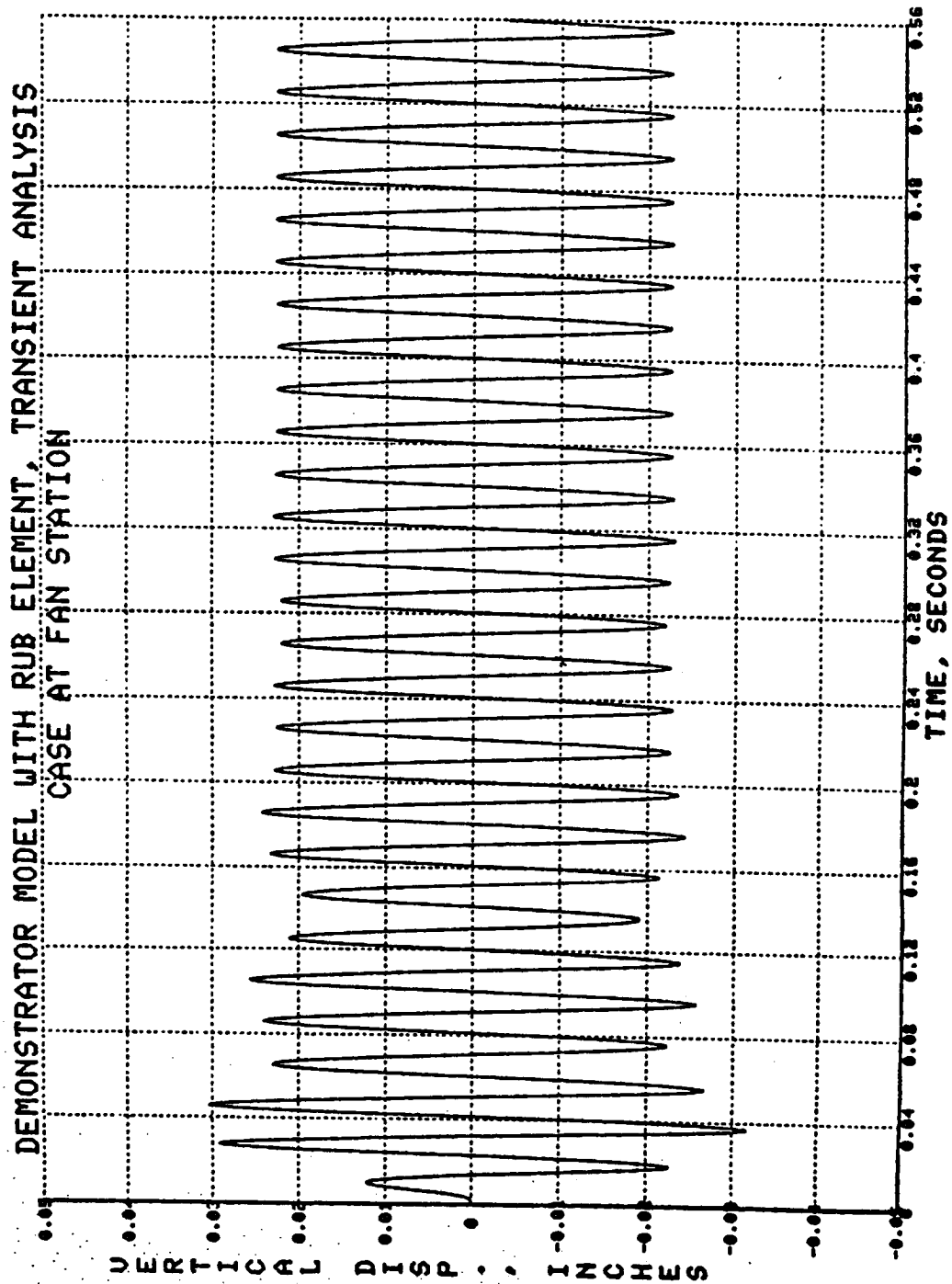


FIGURE 5-28. DEMONSTRATOR TRANSIENT ANALYSIS RESPONSE PLOT

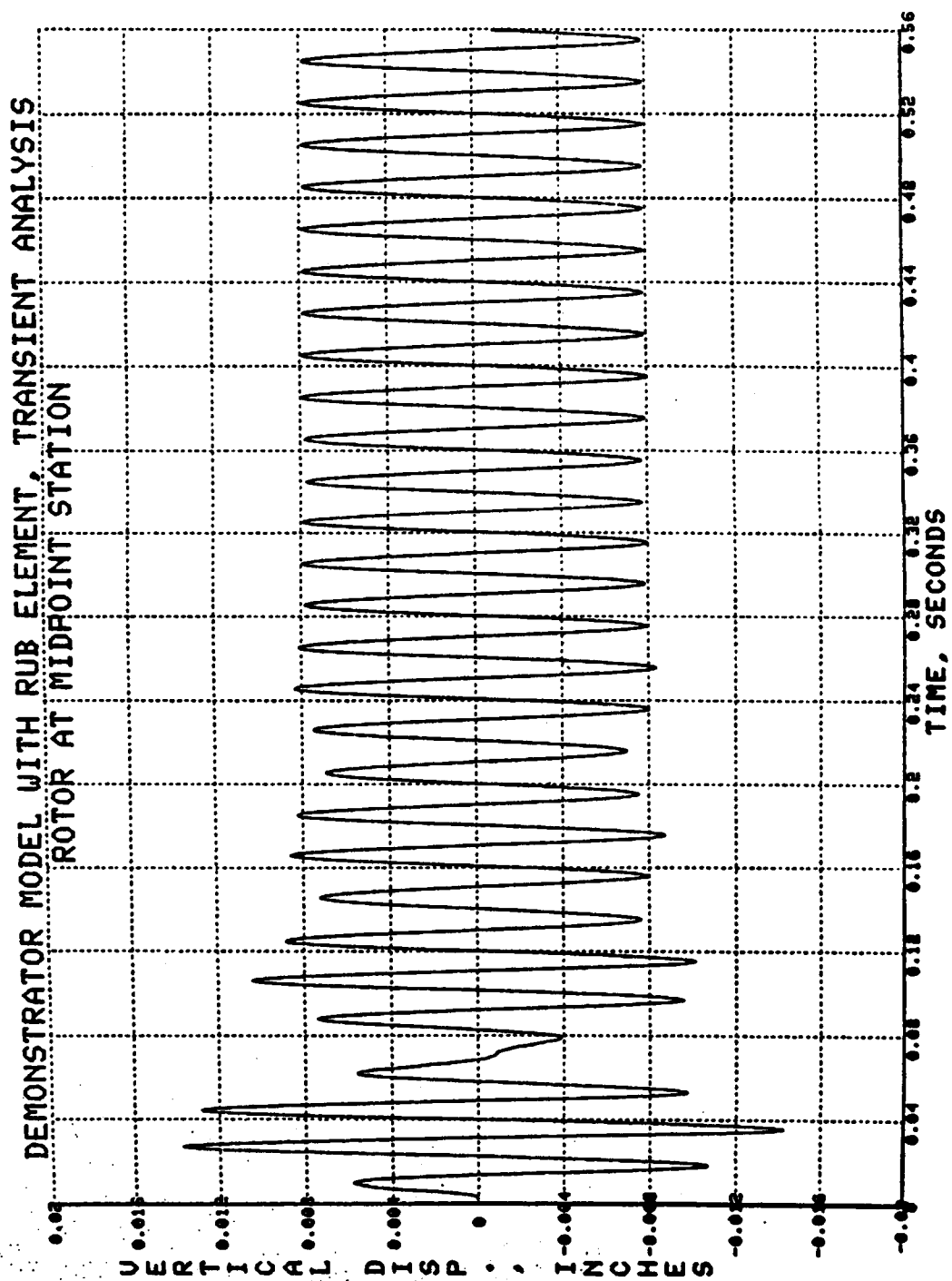


FIGURE 5-29. DEMONSTRATOR TRANSIENT ANALYSIS RESPONSE PLOT



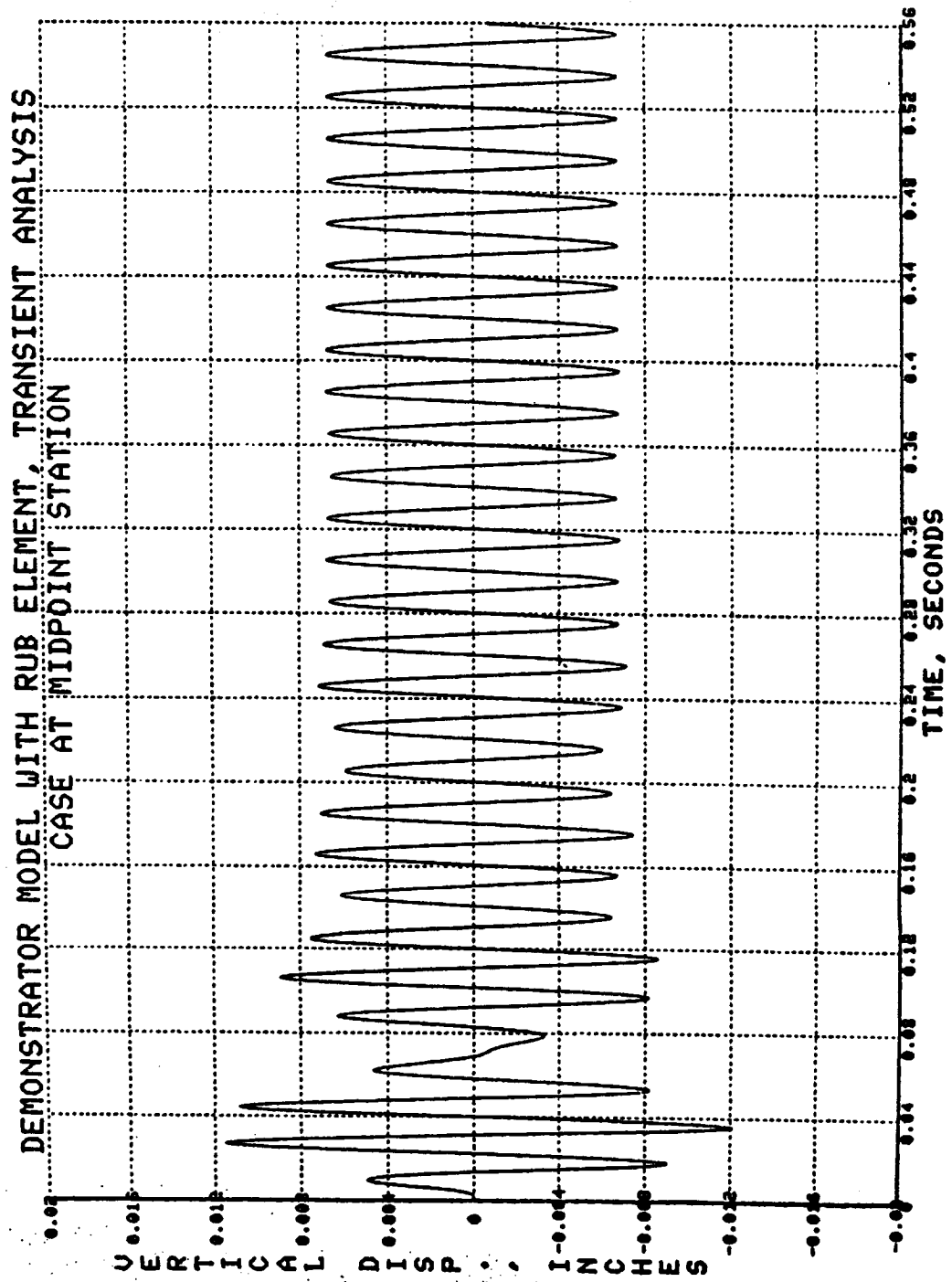


FIGURE 5-30. DEMONSTRATOR TRANSIENT ANALYSIS RESPONSE PLOT

The next orbit plot, figure 5-32, shows only the point on the rotor at the fan station, but for the full 0 through .56 seconds of the transient analysis run. Likewise, figure 5-33 shows only the point on the case at the fan station for the full 0 through .56 seconds of the transient analysis run. Displacements start at 0 at time = 0 (the center of the plot), then there is an overshoot where for just a revolution or two the displacements are appreciable higher than the rest of the run, after which the motion settles into a steady state phase at a constant radius. This is the same pattern exhibited by figures 5-27 through 5-30.

Next, figure 5-34, like figure 5-32, shows only the point on the rotor at the fan station. Likewise, figure 5-35, like figure 5-33, shows only the point on the case at the fan station. However, figures 5-34 and 5-35 were generated from the steady state analysis run, whereas figures 5-32 and 5-33 were generated from the transient analysis run. Since figures 5-34 and 5-35 plot steady state motion only, the transient effects of displacement build-up and overshoot are gone, and what is left is a circular orbital path. By overlaying figure 5-34 on top of figure 5-32, and figure 5-35 on top of figure 5-33, we can see that the steady state analysis results agree very well with the transient analysis results after the transient effect has died out.

Figure 5-36 shows a different type of orbit plot. This plot is for the transient analysis run for the full 0 through .56 seconds. However, this plot shows the relative displacement components for the rub element on the x and y axes, whereas the earlier orbit plots showed the actual physical displacement components of the two joints of the rub element on the x and y axes. (The relative displacement for the rub element is defined as the displacement of joint I of the rub element minus that of joint J). One thing that is neat about this type of orbit plot is that we also show the clearance circle on this plot. The radius of the clearance circle is the dead band which was inputted, in this case .005 inches. As can be seen, the dead band is exceeded (it rubs) for all except the very early portion of the transient analysis run.

Finally, figure 5-37 gives the same type of orbit plot as figure 5-36, except that figure 5-37 was generated from the steady state analysis run, whereas figure 5-36 was generated from the transient analysis run. Again, by overlaying figure 5-37 on top of figure 5-36, we can see that the steady state analysis results agree very well with the transient analysis results after the transient effect has died out.

Note that this case produced circular orbital paths (at least after the transient effect died out). The reason that circular orbital paths are produced is that the input stiffnesses of the physical connecting elements are the same in the vertical and horizontal directions, so neither direction has a tendency to displace more than the other under the influence of the unbalance load. Incidentally, the cases discussed in sections 5.3.3.1 and 5.3.3.2 also had input stiffnesses that were the same in the vertical and horizontal directions, and those cases would also produce circular orbital plots similar to the ones shown in this section.

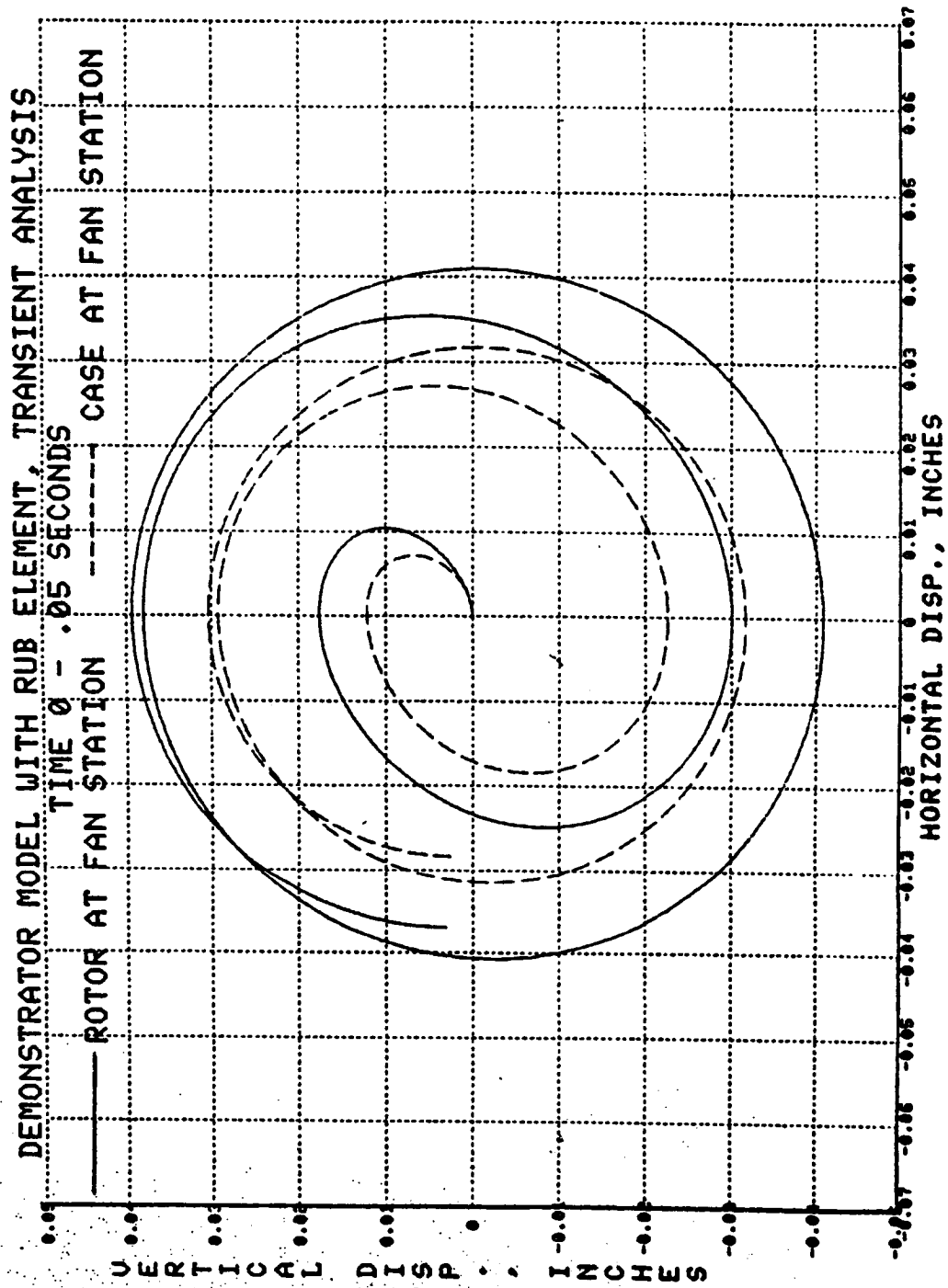


FIGURE 5-31. DEMONSTRATOR TRANSIENT ANALYSIS ORBIT PLOT

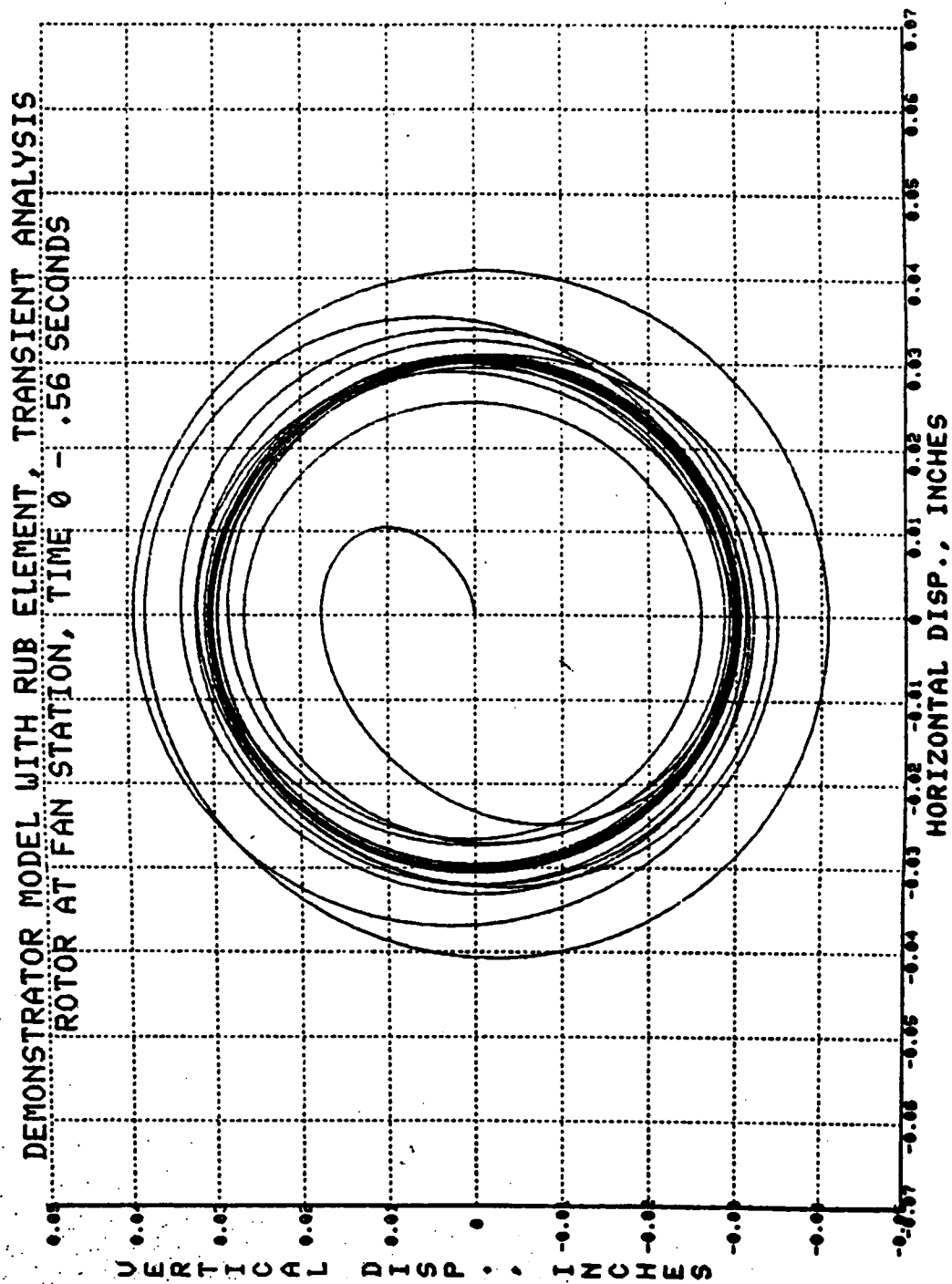


FIGURE 5-32. DEMONSTRATOR TRANSIENT ANALYSIS ORBIT PLOT

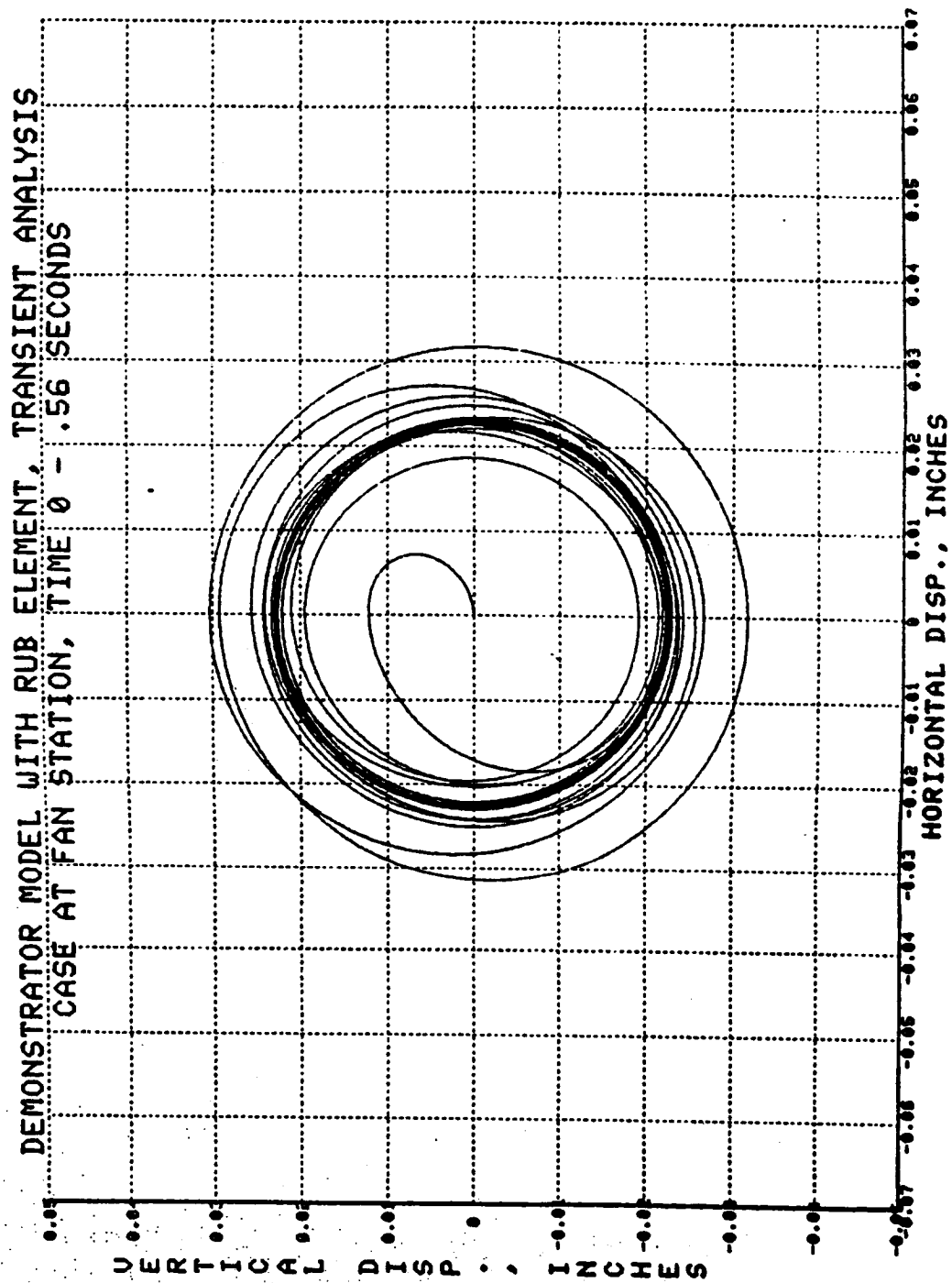


FIGURE 5-33. DEMONSTRATOR TRANSIENT ANALYSIS ORBIT PLOT

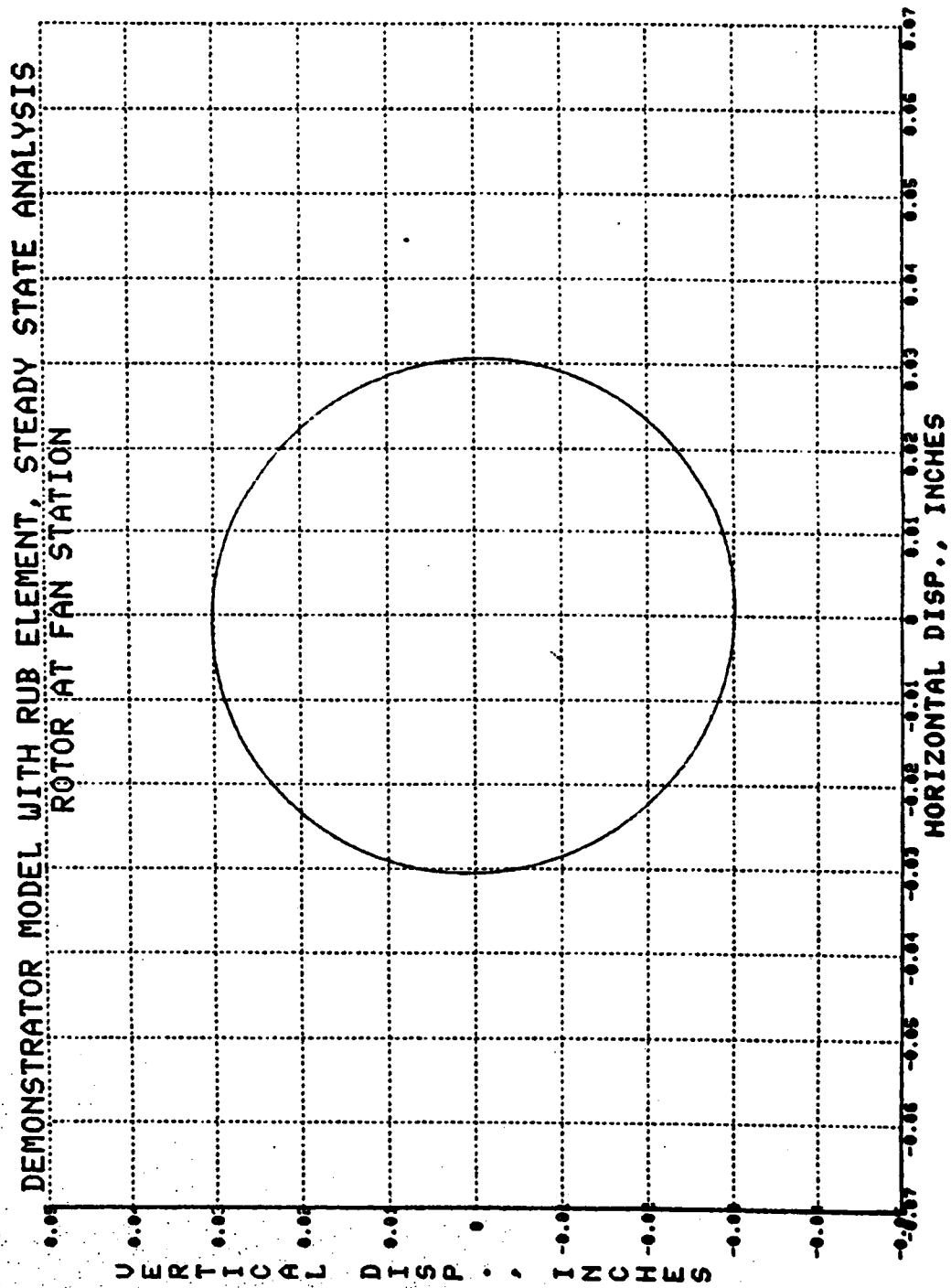


FIGURE 5-34. DEMONSTRATOR STEADY STATE ANALYSIS ORBIT PLOT

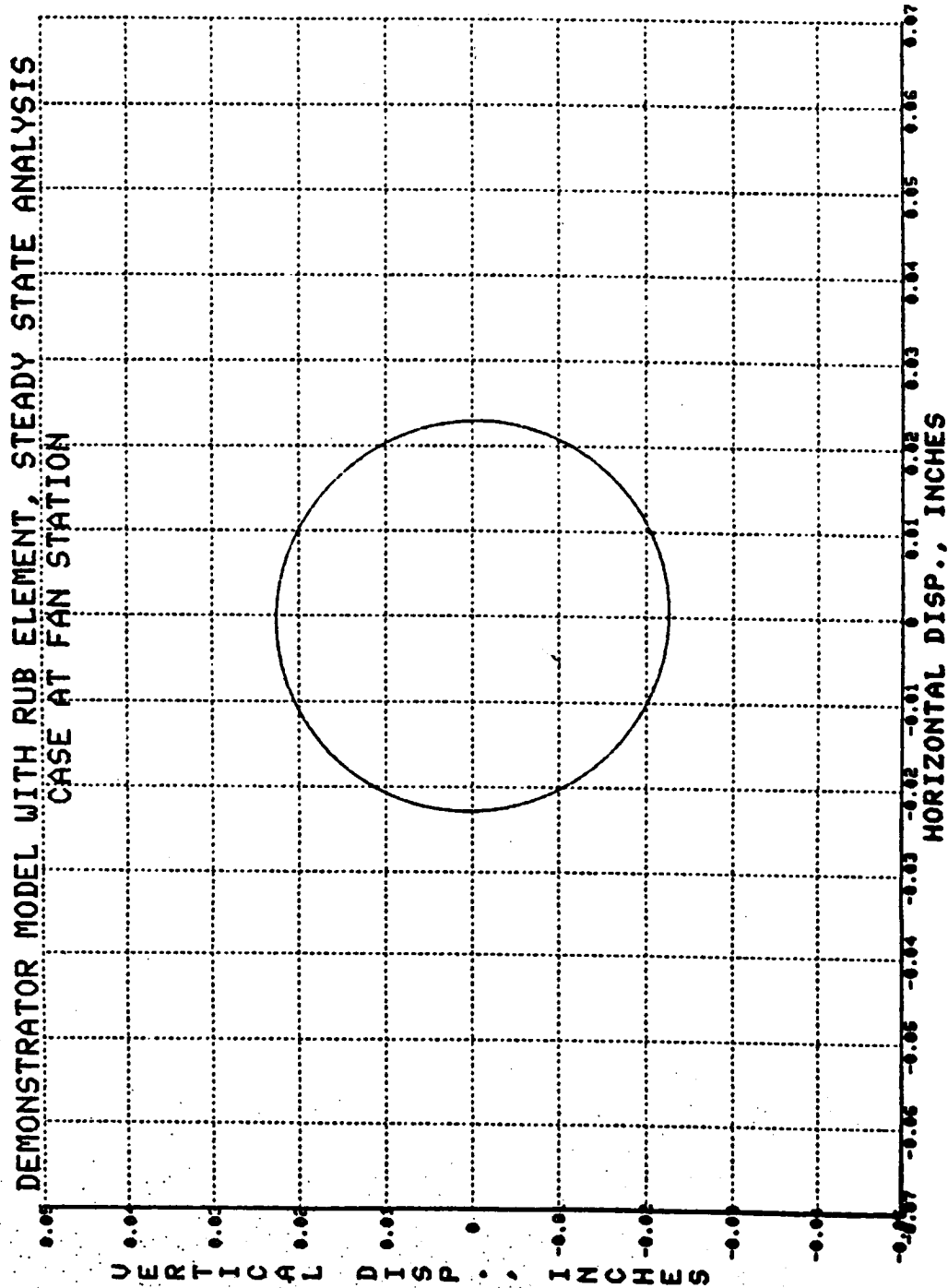


FIGURE 5-35. DEMONSTRATOR STEADY STATE ANALYSIS ORBIT PLOT

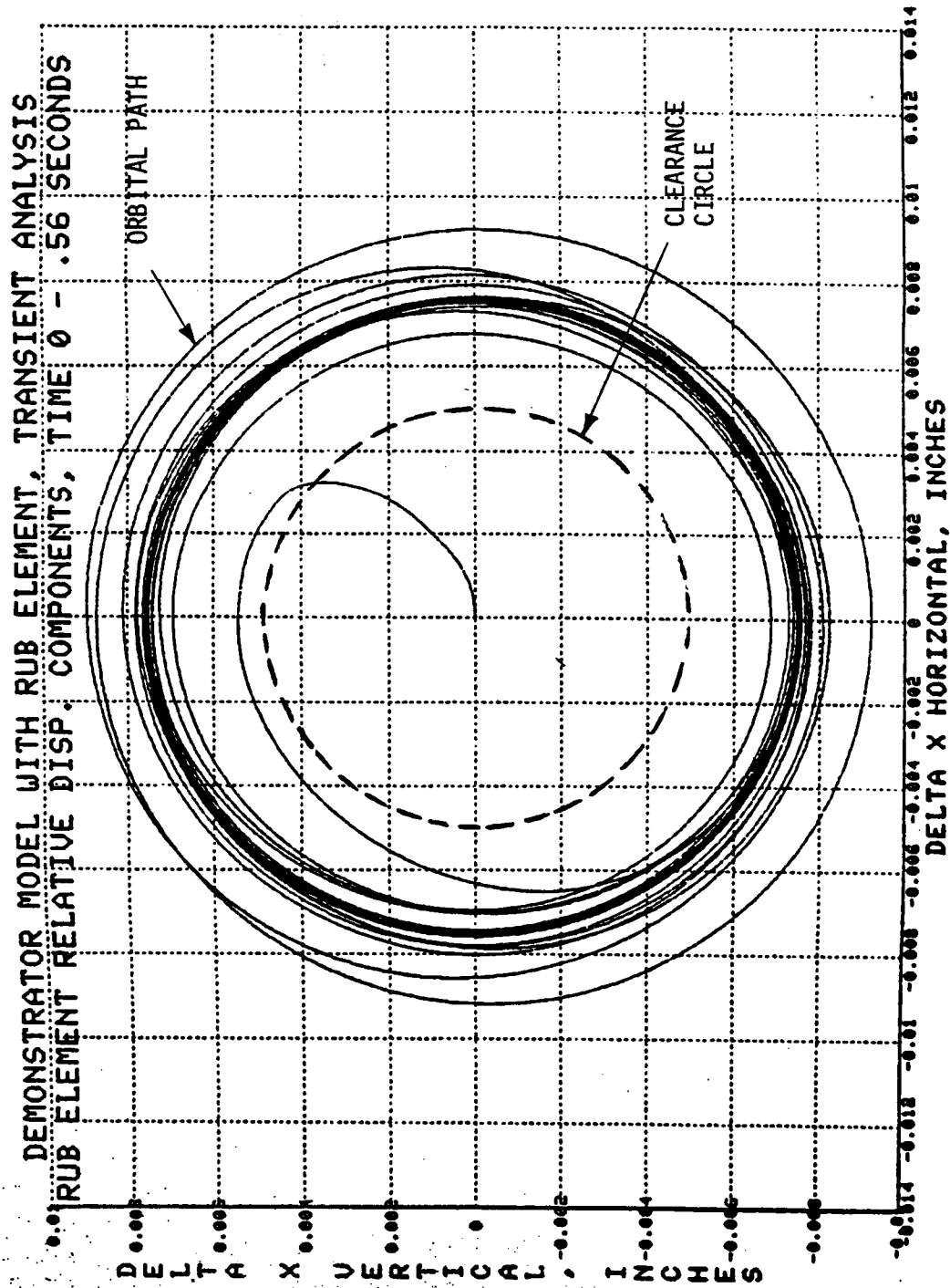


FIGURE 5-36. DEMONSTRATOR TRANSIENT ANALYSIS ORBIT PLOT



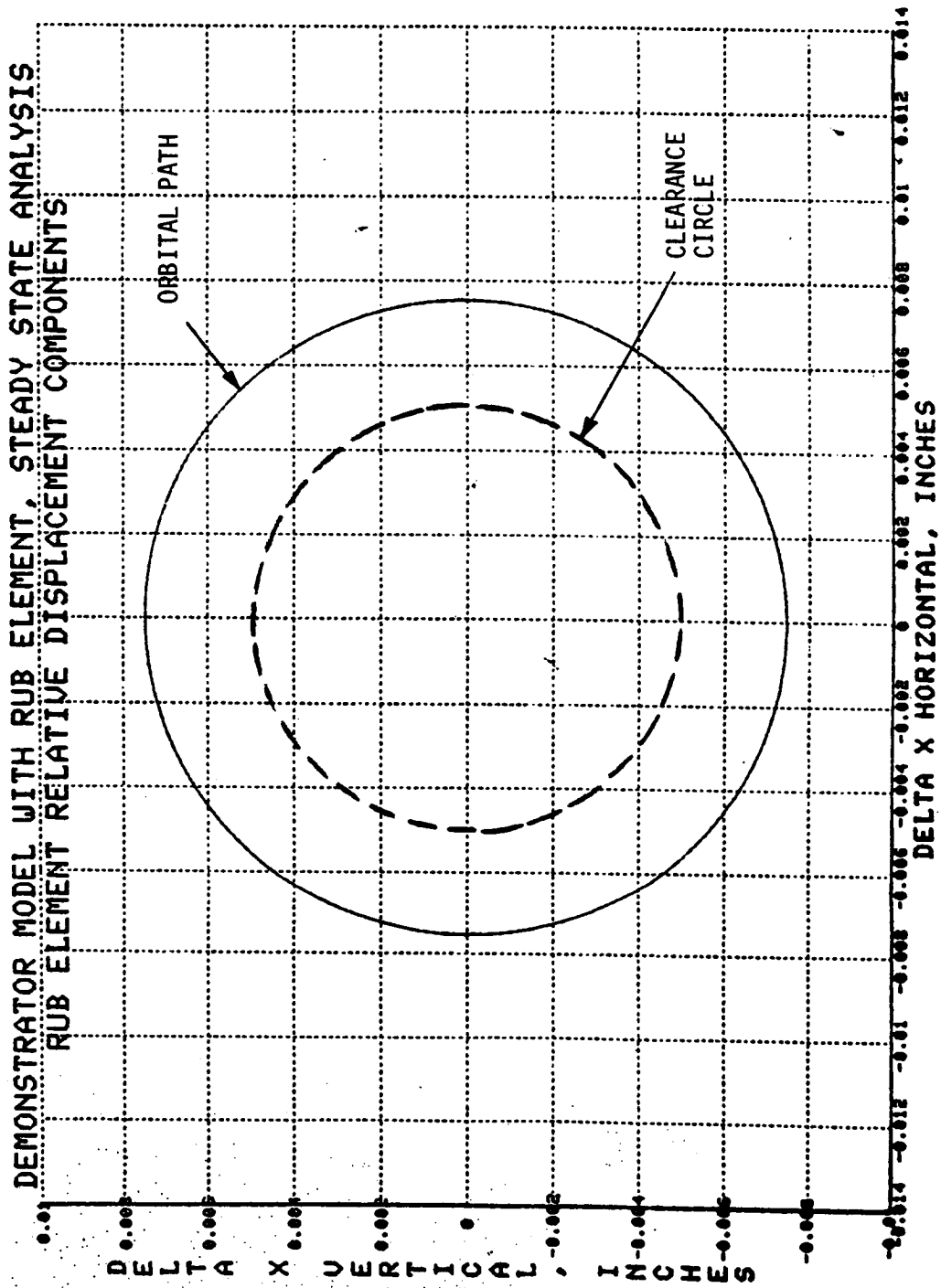


FIGURE 5-37. DEMONSTRATOR STEADY STATE ANALYSIS ORBIT PLOT

#### 5.3.3.4 Intermittent Rub

Finally, a steady state analysis test case was run where an intermittent rub resulted. An intermittent rub occurs less frequently than no rub or a continual rub, and the input had to be adjusted carefully to make it happen. The previous demonstrator test cases had the same spring constants in the vertical and horizontal directions. However, this results in circular orbital paths (see figures 5-31 through 5-37). Circular orbital paths will not cause an intermittent rub, because the clearance circle is also circular, and a circular orbital path will either lie within the clearance circle (no rub) or outside the clearance circle (continual rub).

To cause an intermittent rub, the mount springs for the casing (springs 1 and 2 in figure 5-9) were made 5 times more stiff in the vertical direction than in the horizontal direction. Stiffer casing springs in the vertical direction than the horizontal direction might result, for instance, from an engine attached to a wing via a pylon. This discrepancy in spring rates in the two directions forced the orbital path to take an elliptical shape. The input dead band was then adjusted so that the orbital ellipse intersected the clearance circle at four points, causing an intermittent rub, as illustrated in figure 5-18.

As for the previous test cases for the demonstrator model with one rub element, a transient analysis run taken to steady state conditions was made for comparison. The transient analysis results are shown in figures 5-38 through 5-45. The transient analysis run was stopped at .56 seconds, by which time the transient analysis effect had almost died out, as can be seen from these plots. Plots for the horizontal direction displacements are included as well as those for the vertical. Only the vertical direction displacement plots were included for the test cases described in sections 5.3.3.1, 5.3.3.2, and 5.3.3.3, because the vertical and horizontal displacement plots were essentially the same for those test cases due to the circular orbital paths.

The results are summarized in table 5-4 on page 191. For the test cases described in sections 5.3.3.1, 5.3.3.2, and 5.3.3.3, the maximum horizontal displacements and maximum vertical displacements were the same since the orbital paths were circular. Therefore, only the maximum vertical displacement results were tabulated for those test cases. However, for the intermittent rub test case the orbital path is elliptical, and the maximum horizontal displacements are much larger than the maximum vertical displacements. Therefore, both vertical and the horizontal displacement results are summarized in table 5-4. The steady state analysis values in this table are exact numbers from the printout of the run. The transient analysis run numbers, however, had to be estimated by reading the peak values in the steady state region (.4 to .56 seconds) of figures 5-38 through 5-45.

This test case provides the most difficult challenge for the steady state analysis to meet. As is seen from table 5-4, agreement between the steady state and transient analysis runs is very good for most point and directions. However, agreement is not as good as for the test cases of sections 5.3.3.1, 5.3.3.2, and 5.3.3.3, and agreement is poor for one point and direction (midpoint of rotor in

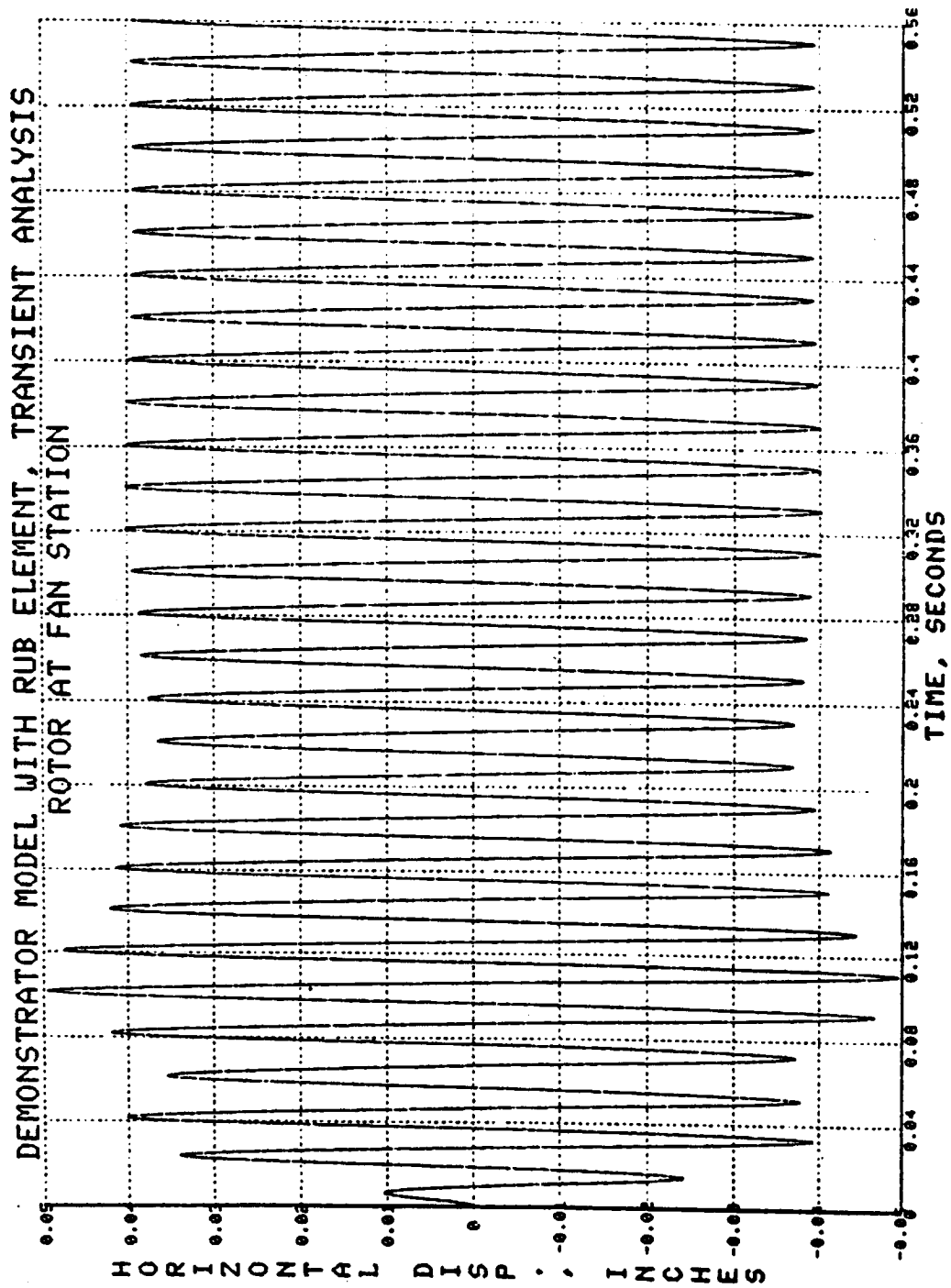


Figure 5-38. Demonstrator Transient Analysis Response Plot

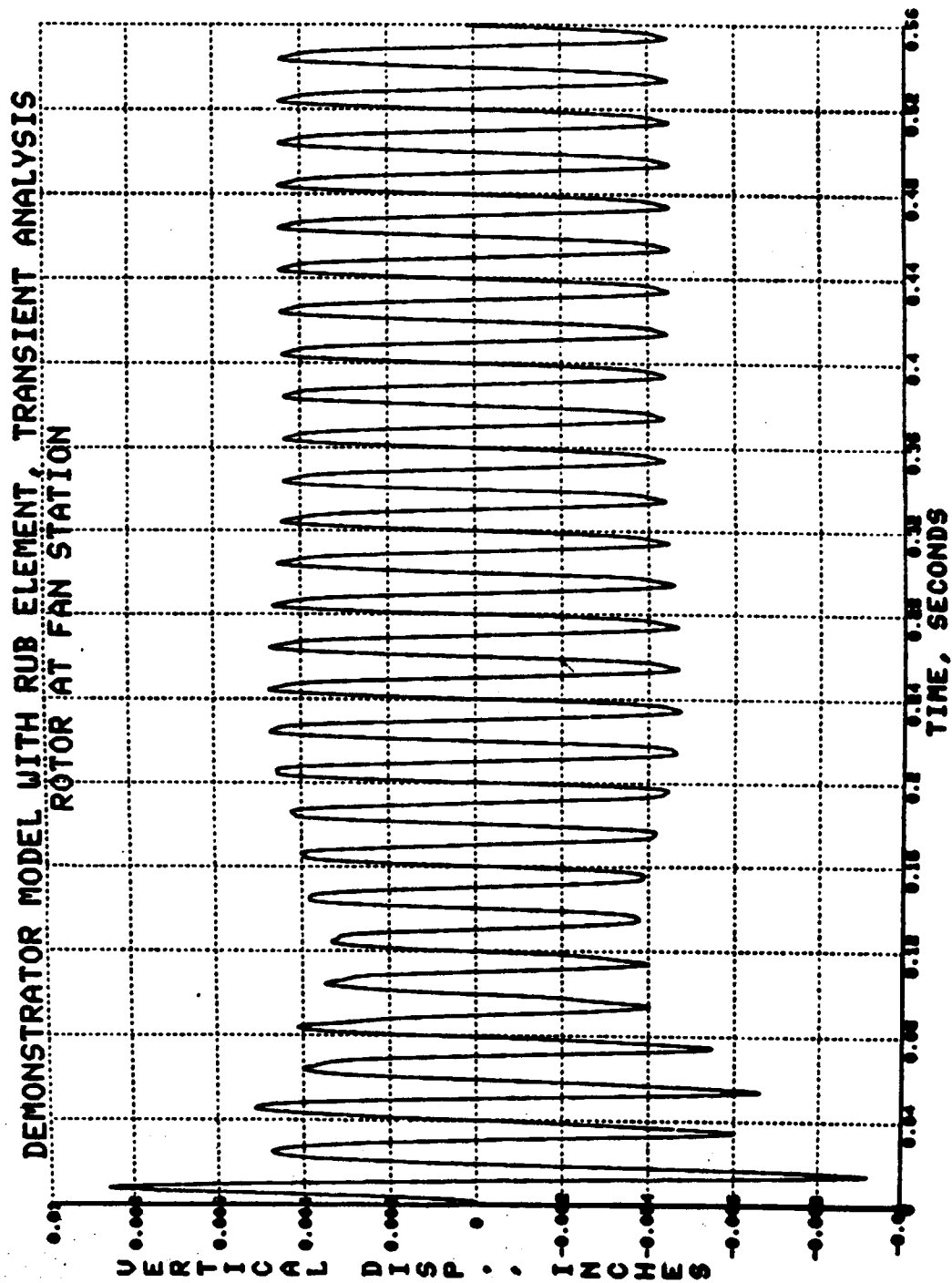


Figure 5-39. Demonstrator Transient Analysis Response Plot

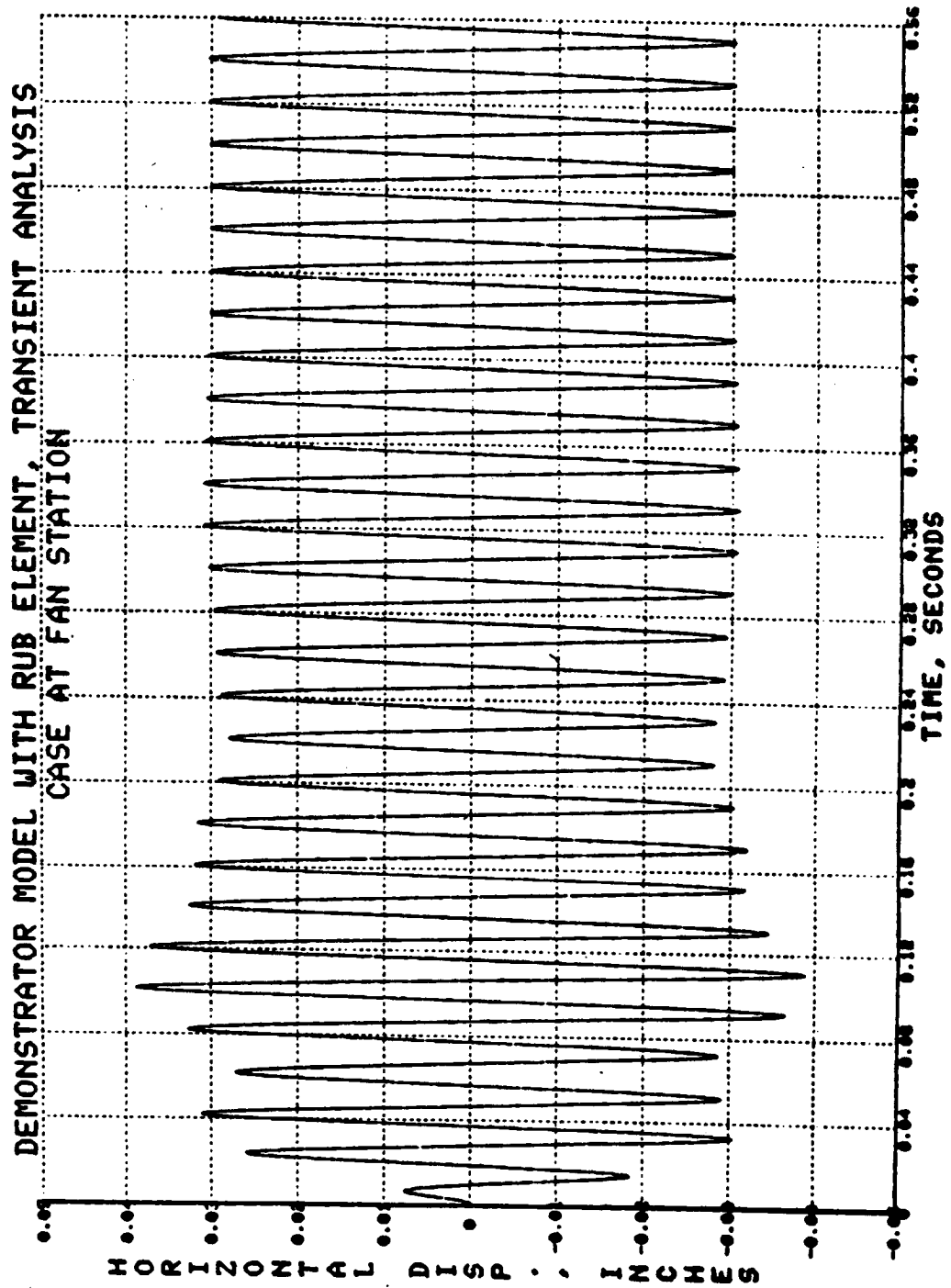


Figure 5-40. Demonstrator Transient Analysis Response Plot

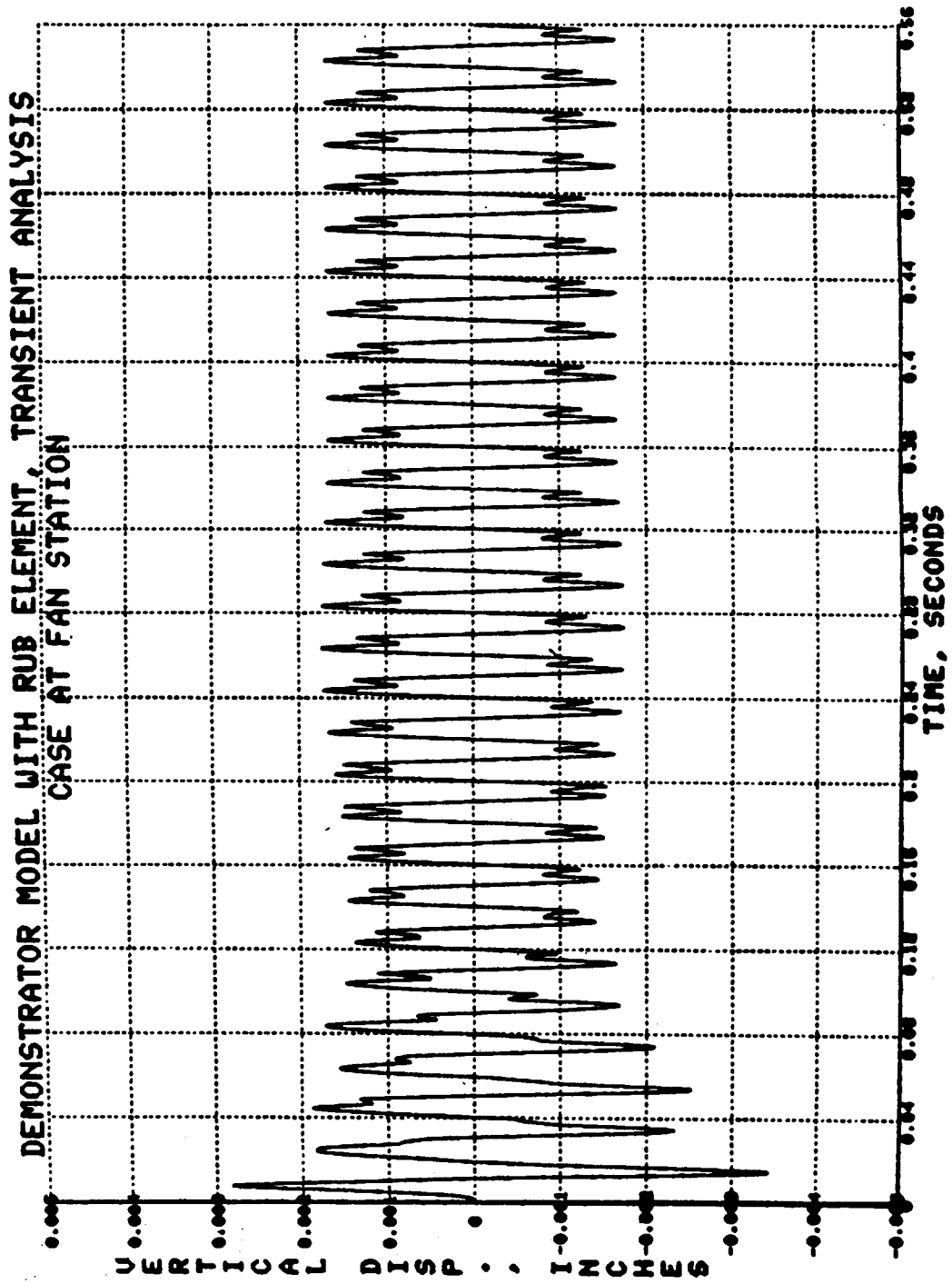


Figure 5-41. Demonstrator Transient Analysis Response Plot

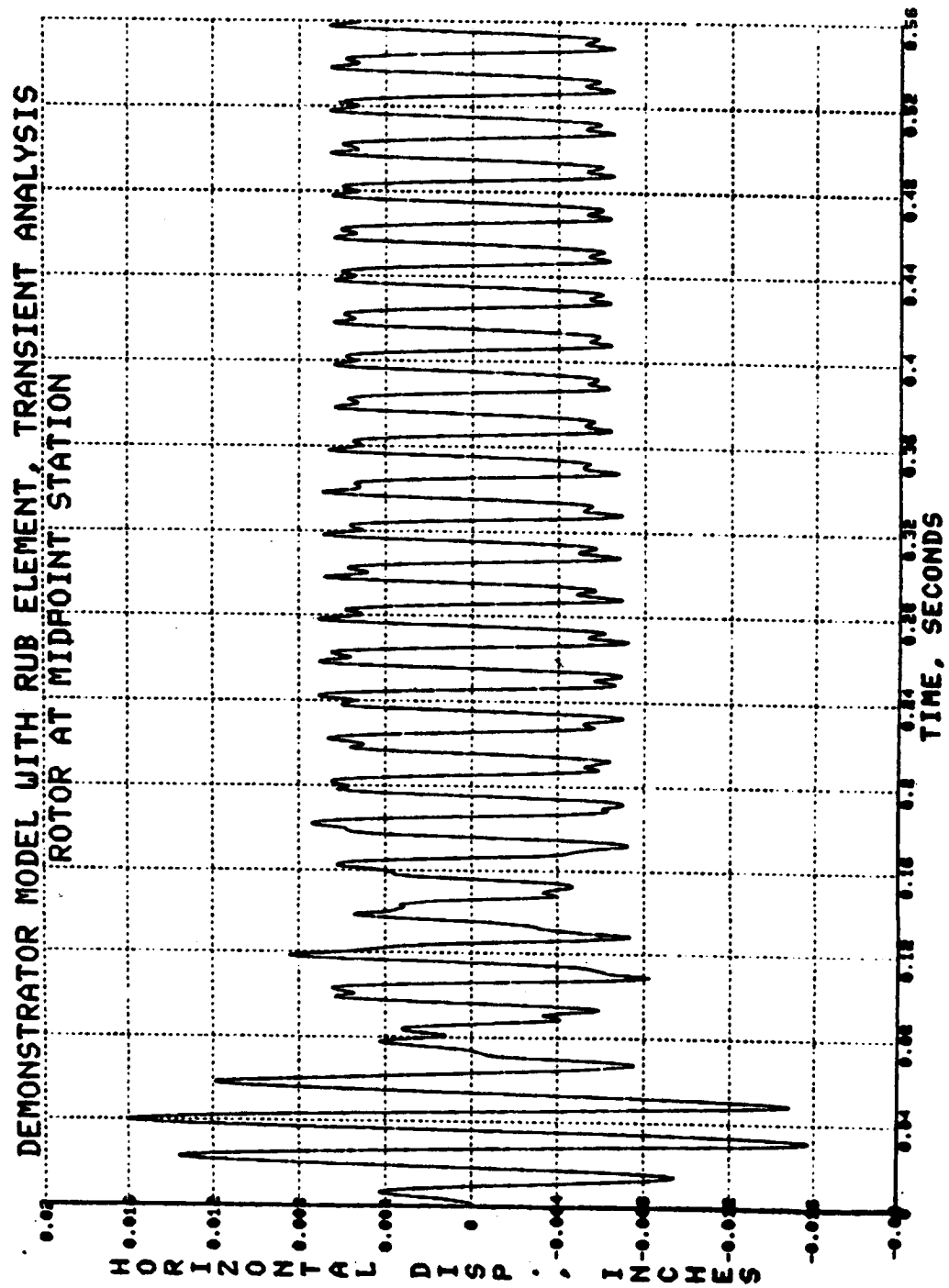


Figure 5-42. Demonstrator Transient Analysis Response Plot

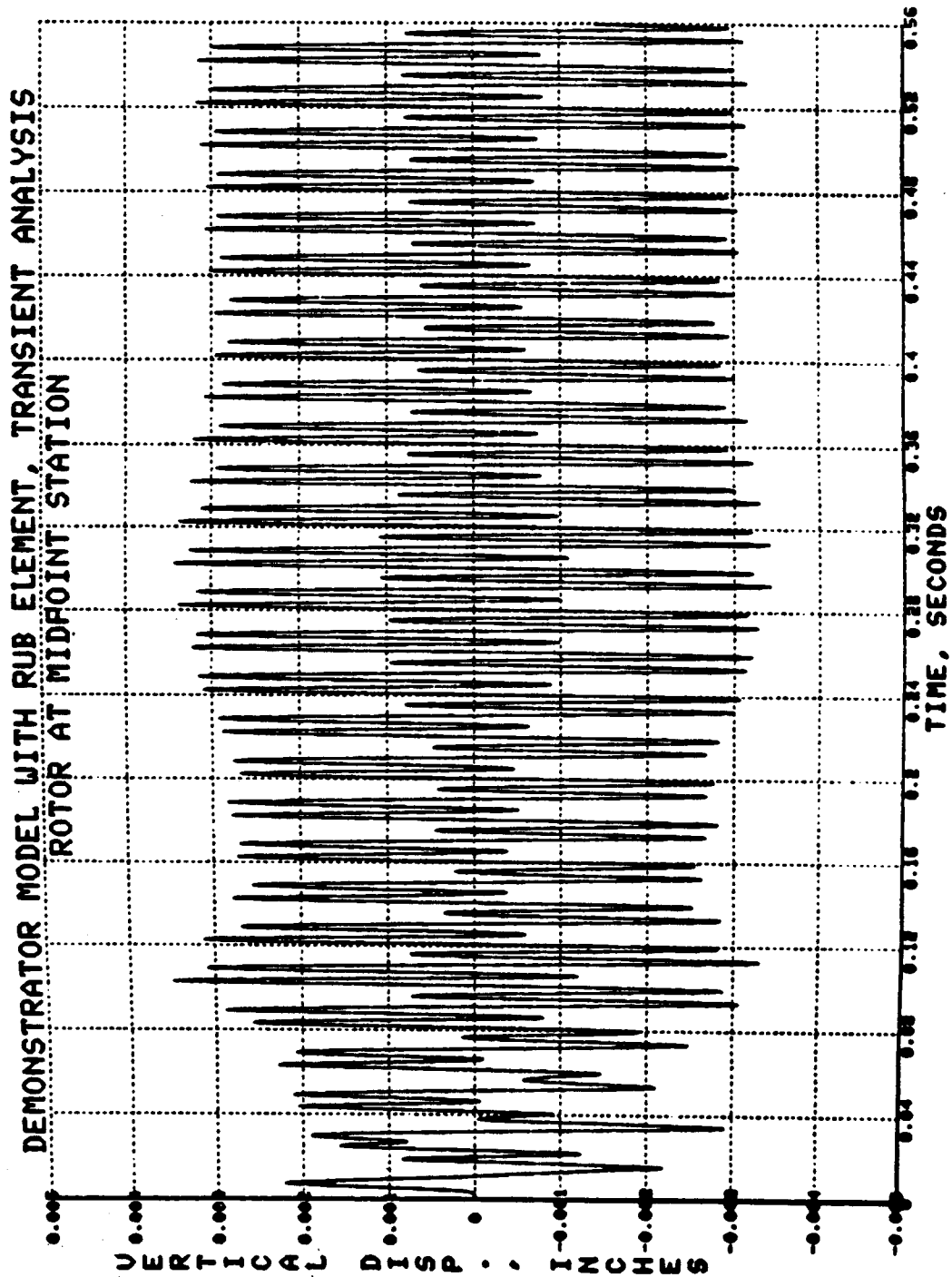


Figure 5-43. Demonstrator Transient Analysis Response Plot



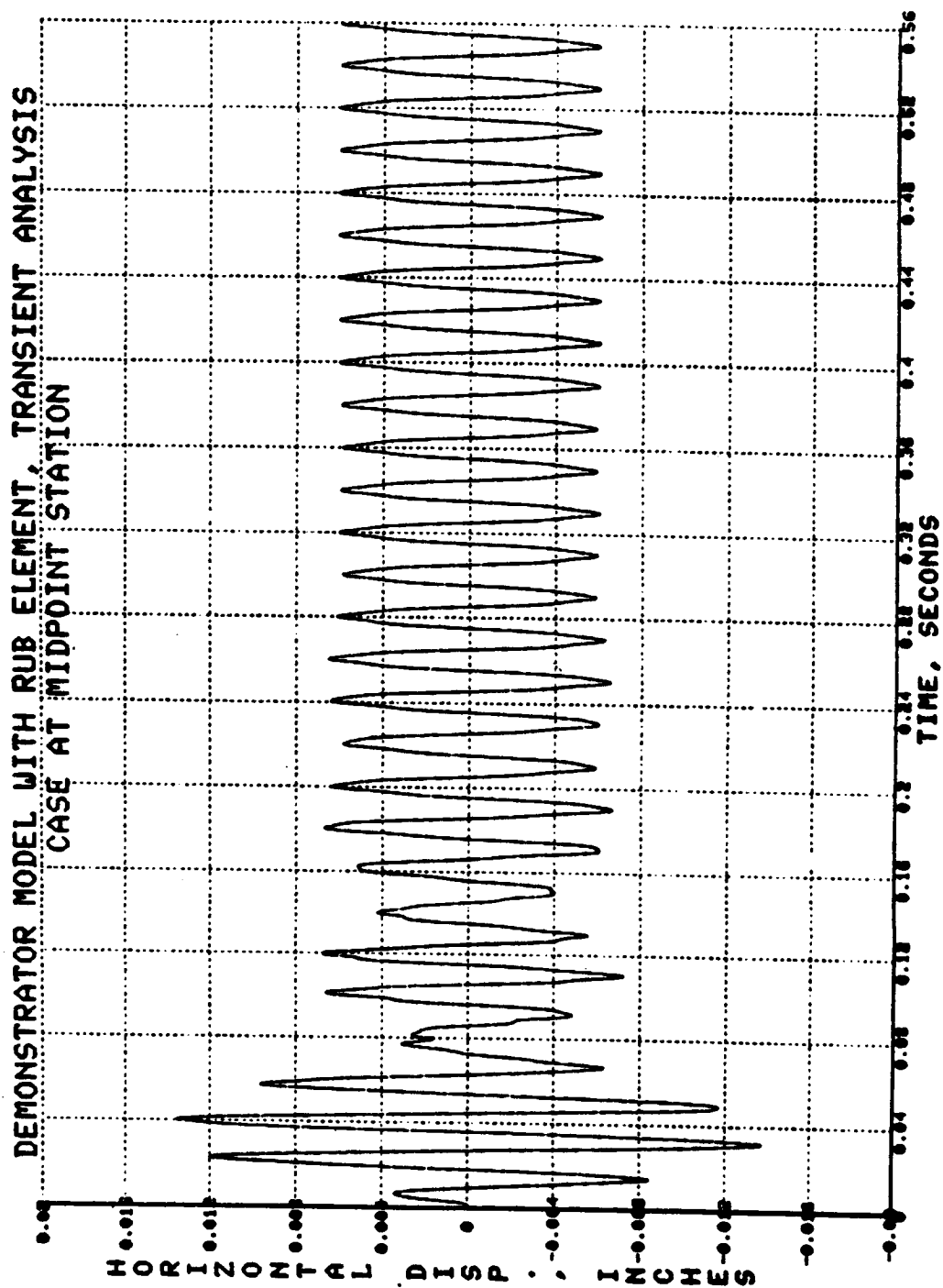


Figure 5-44. Demonstrator Transient Analysis Response Plot

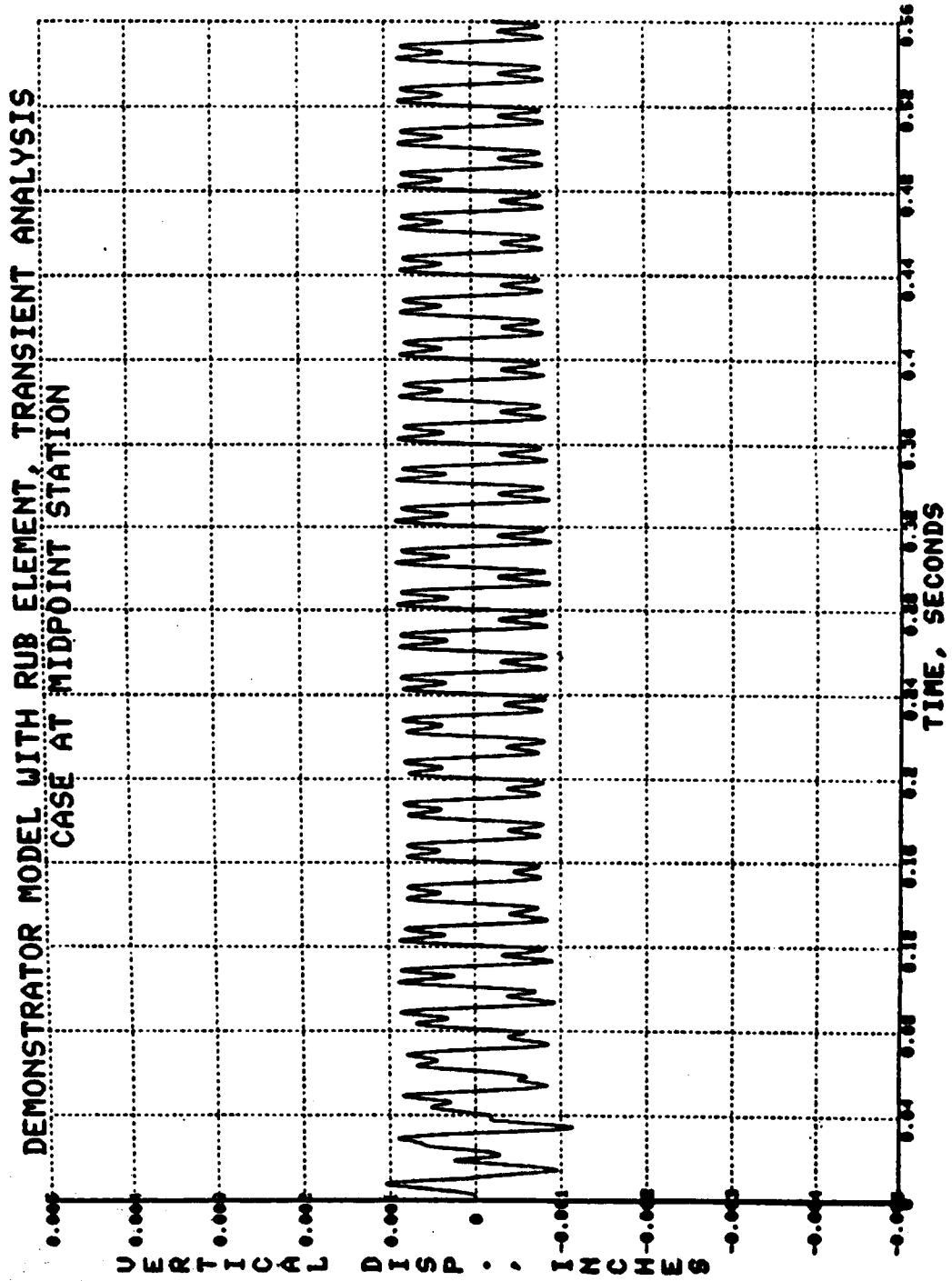


Figure 5-45. Demonstrator Transient Analysis Response Plot

the vertical direction).

Table 5-4. Demonstrator Model Results with Intermittent Rub

<u>Location</u>	<u>Direction</u>	<u>Maximum Displacement (inches)</u>		<u>Transient</u>
		<u>Steady State</u>	<u>Transient</u>	<u>Analysis</u>
		<u>Analysis Run</u>	<u>Analysis Run</u>	<u>Figure</u>
				<u>Number</u>
Rotor at fan station	Horizontal	.03905	.039	5-38
Rotor at fan station	Vertical	.00479	.0045	5-39
Case at fan station	Horizontal	.02961	.030	5-40
Case at fan station	Vertical	.00149	.0017	5-41
Rotor at midpoint station	Horizontal	.00698	.0067	5-42
Rotor at midpoint station	Vertical	.00148	.0030	5-43
Case at midpoint station	Horizontal	.00596	.0060	5-44
Case at midpoint station	Vertical	.00076	.0008	5-45

The reason that this test case does not agree as well as the previous test cases has to do with the interaction between the rotor and case resulting from the rub element. For a transient analysis run, the rub element works as follows. For a good portion of the orbit, the clearance circle is not exceeded, and the rub element provides no force. However, twice during each orbit the clearance circle is exceeded and a rub element force is exerted on the rotor and case. This on/off rub force has a complex effect on the vibration of the rotor and the case. Rather than just first harmonic vibration, involving the terms  $\cos\omega t$  and  $\sin\omega t$ , higher order terms involving  $\cos 2\omega t$ ,  $\sin 2\omega t$ ,  $\cos 3\omega t$ ,  $\sin 3\omega t$ , etc. may be produced.

Looking at the transient analysis response plots for this case, figures 5-38, 5-39, 5-40, and 5-44 appear to involve only simple first harmonic vibration, with a frequency of vibration  $\omega$  equalling the forcing frequency, which in this case is the rotor speed (3000 rpm). However, in figures 5-41, 5-42, 5-43, and 5-45, two frequencies of vibration are observed - the forcing frequency of 3000 rpm and also three times that (9000 rpm). So, in addition to first harmonic vibration, involving the terms  $\cos\omega t$  and  $\sin\omega t$ , we have third harmonic vibration, involving the terms  $\cos 3\omega t$  and  $\sin 3\omega t$ .

Steady state analyses are different. For steady state analyses, the technique of harmonic averaging is used to find the rub element forces, as explained in section 5.2 of volume 1. Rather than apply the rub force in an on/off manner,

for steady state analyses the rub force is smeared over the entire orbit. All higher order terms are neglected as part of harmonic averaging, leaving only the first harmonic terms. For this reason, for intermittent rubs the transient analysis taken to steady state conditions provides a more accurate representation of the true characteristics than does the steady analysis.

This is also the reason for the poor agreement between the steady state and transient analyses for the rotor at the midpoint station in the vertical direction. The transient analysis response plot for the rotor at the midpoint station in the vertical direction (figure 5-43) shows that the 9000 rpm mode (the third harmonic mode) is very active for this particular point and direction. Herein lies the rub (pardon the pun), since this mode is completely neglected in the steady state analysis. This was the only one of the eight transient analysis response plots where this mode was so very strong compared to the first harmonic mode, which explains why this point and direction was the only one to have poor agreement. Note that this point and direction are of very little importance, since the vibration levels are small compared to the vibration in the horizontal direction.

Orbit plots for this case are included in figures 5-46 through 5-52. These orbit plots are similar to the ones in section 5.3.3.3, except that the orbital paths are elliptical in shape rather than circular. Section 5.3.3.3 discusses the characteristics of the orbit plots in detail.

The steady state analysis orbital paths for this case agree very well with the transient analysis orbital paths after transient effects have died out, as can be seen by comparing figure 5-49 with figure 5-47, figure 5-50 with figure 5-48, and figure 5-52 with figure 5-51. One difference, however, is that the steady state analysis orbital paths are true ellipses, whereas the transient analysis orbital paths after the transient effect has died out are somewhat distorted ellipses. This distortion results from the complex interaction between the rotor and case transmitted through the rub element, with the rub element force being activated when the orbit exceeds the clearance circle, and deactivated when inside the clearance circle. The transient analysis orbit plots, of course, are more representative of the true orbital paths, because the steady state analysis is an idealization which does not account for the full complexity of the rotor and case interaction.

In summary, excellent results were obtained in verifying the steady state analysis option of the TETRA 2 computer program. Exact agreement between TETRA 2 and closed form equations was obtained for simple linear one and two degree-of-freedom cases. Also, for simple nonlinear one and two degree-of-freedom test cases (with one and two rub elements, respectively), excellent agreement was obtained between TETRA 2 and small computer programs which were written independently. Using the demonstrator model, very good agreement was obtained between TETRA 2 and the NOVAS computer program for a frequency sweep with no rub elements. Finally, for the demonstrator model with one rub element, very good agreement between the steady state and transient analysis options of TETRA 2 was obtained for each of four possible rub categories.

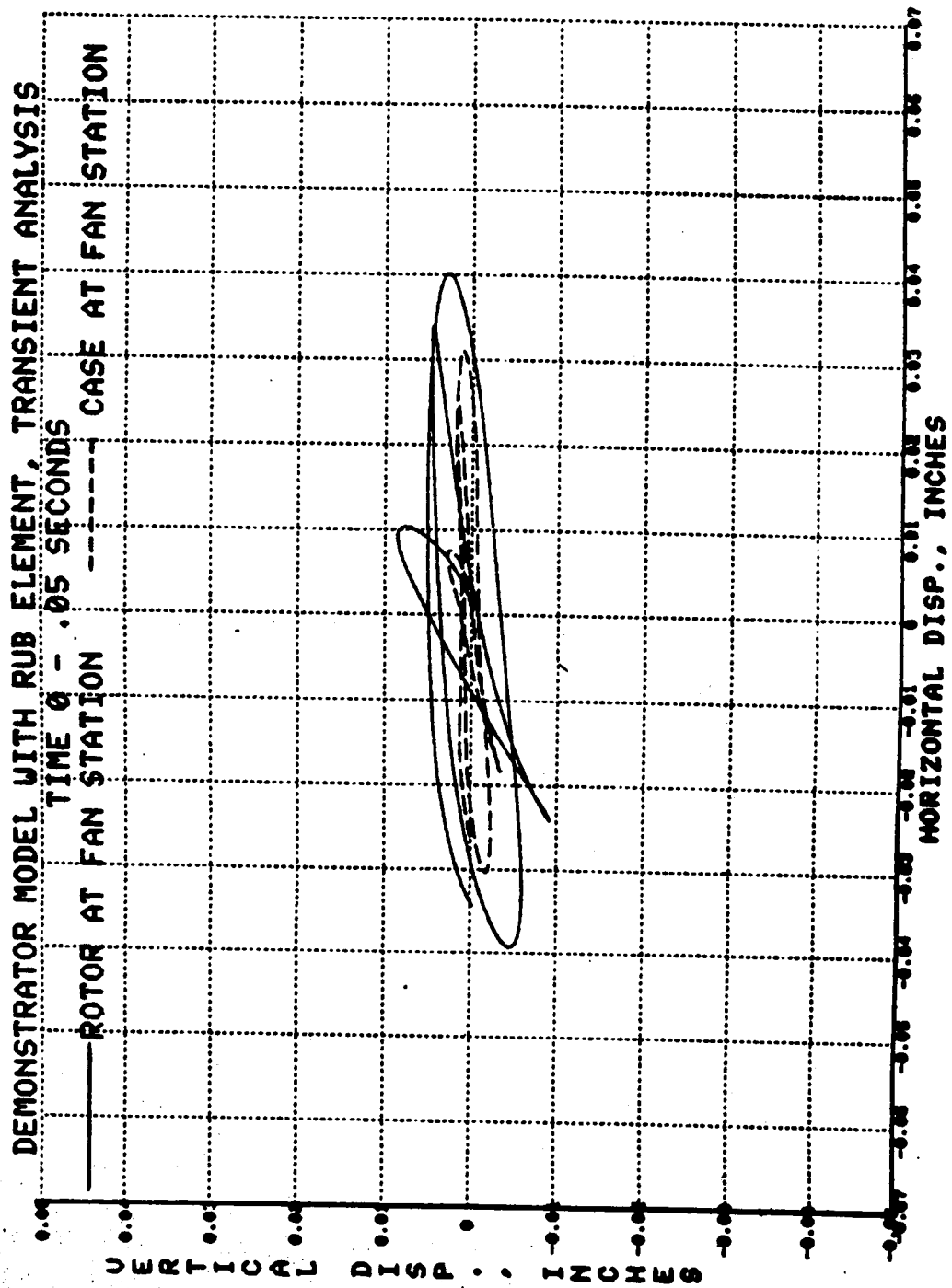


Figure 5-46. Demonstrator Transient Analysis Orbit Plot

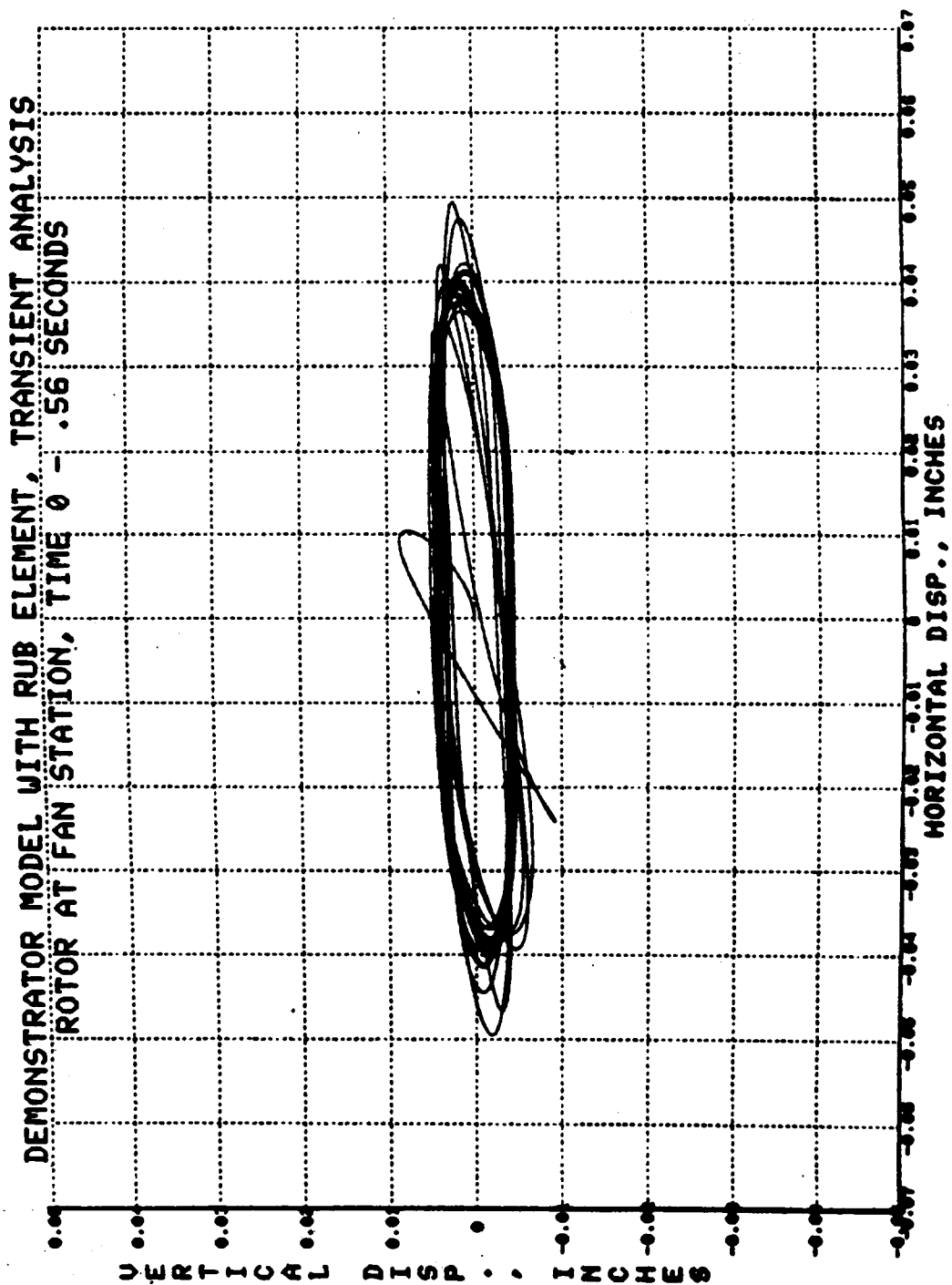


Figure 5-47. Demonstrator Transient Analysis Orbit Plot

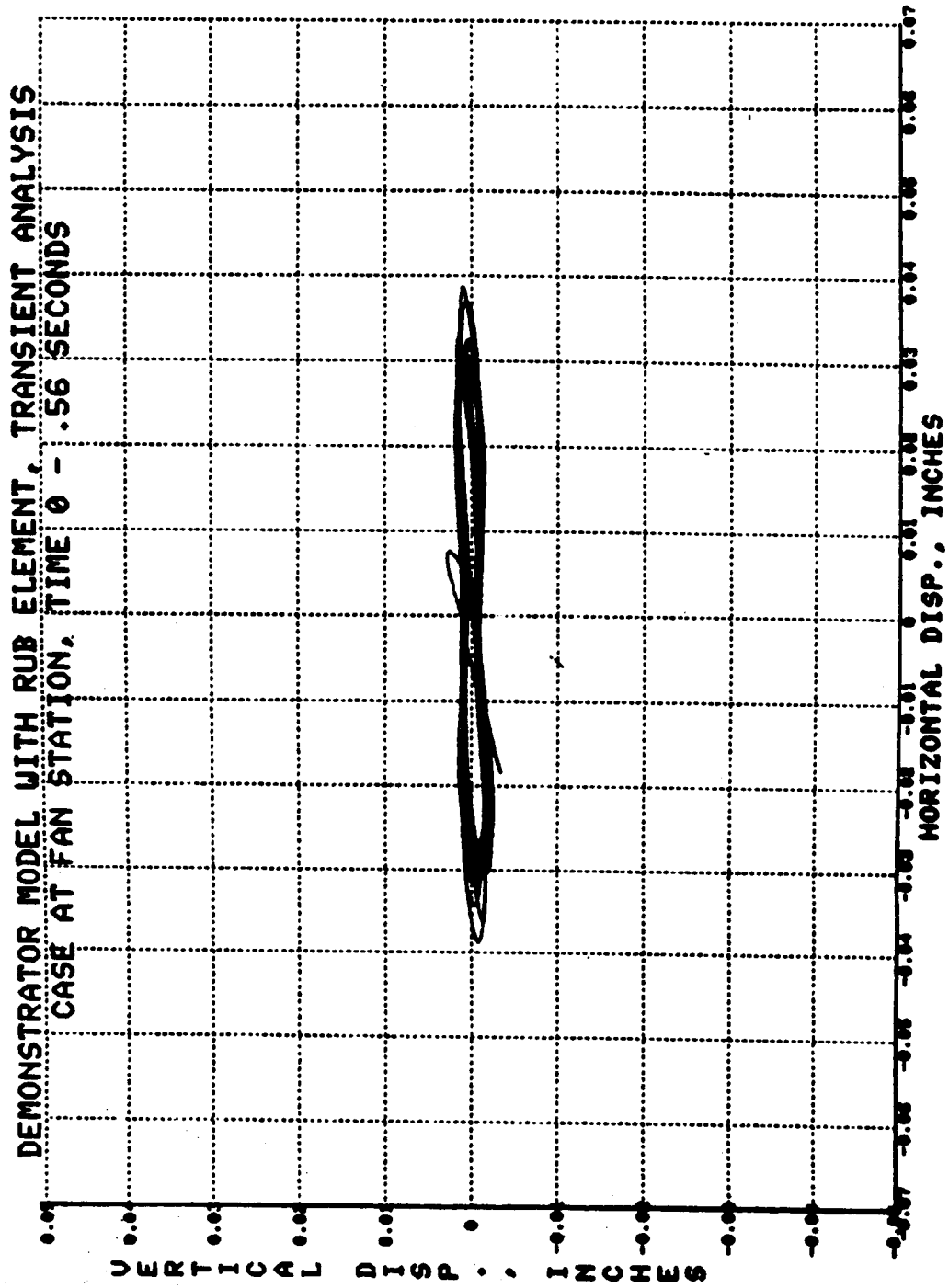


Figure 5-48. Demonstrator Transient Analysis Orbit Plot

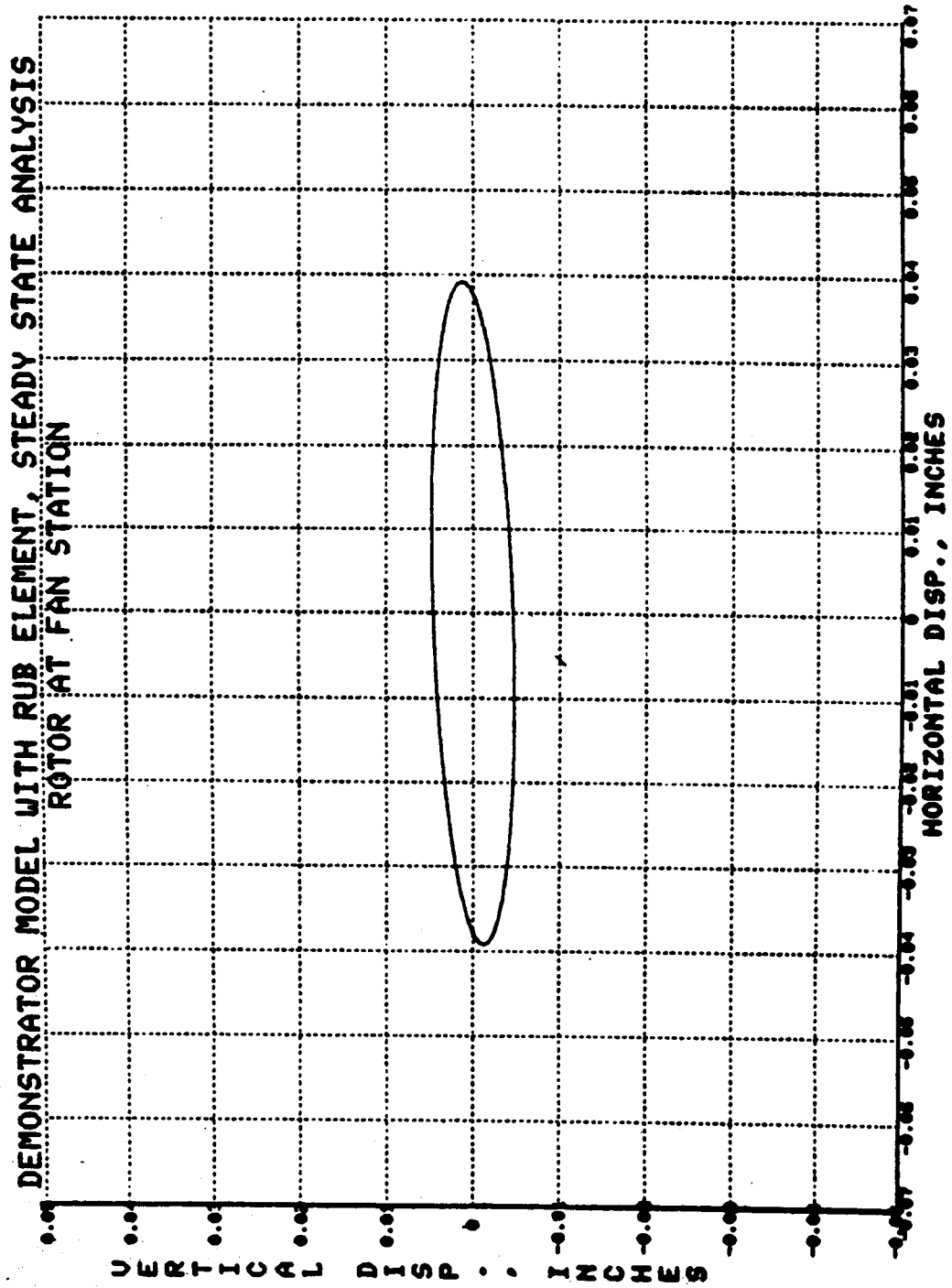


Figure 5-49. Demonstrator Steady State Analysis Orbit Plot



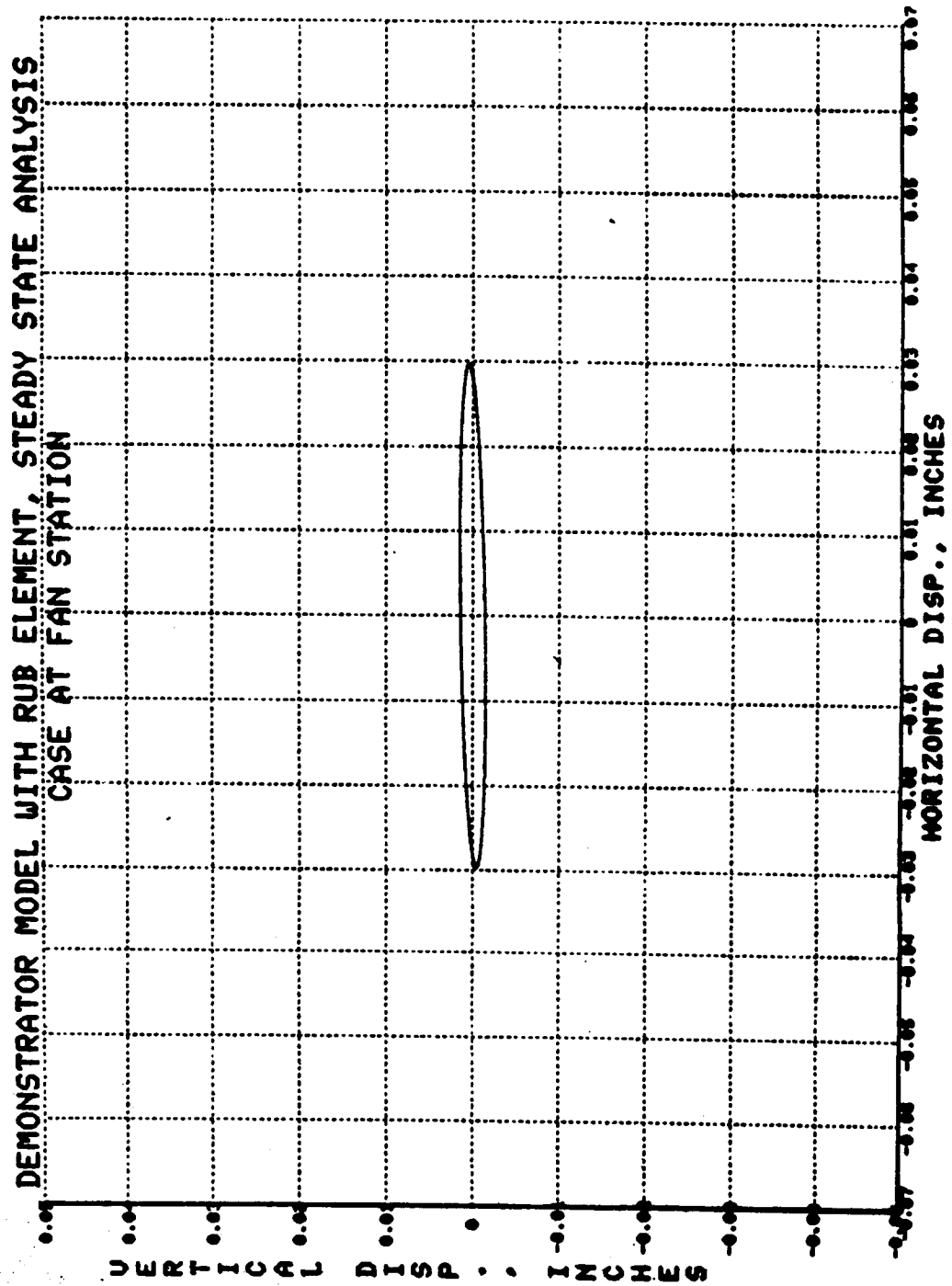


Figure 5-50. Demonstrator Steady State Analysis Orbit Plot

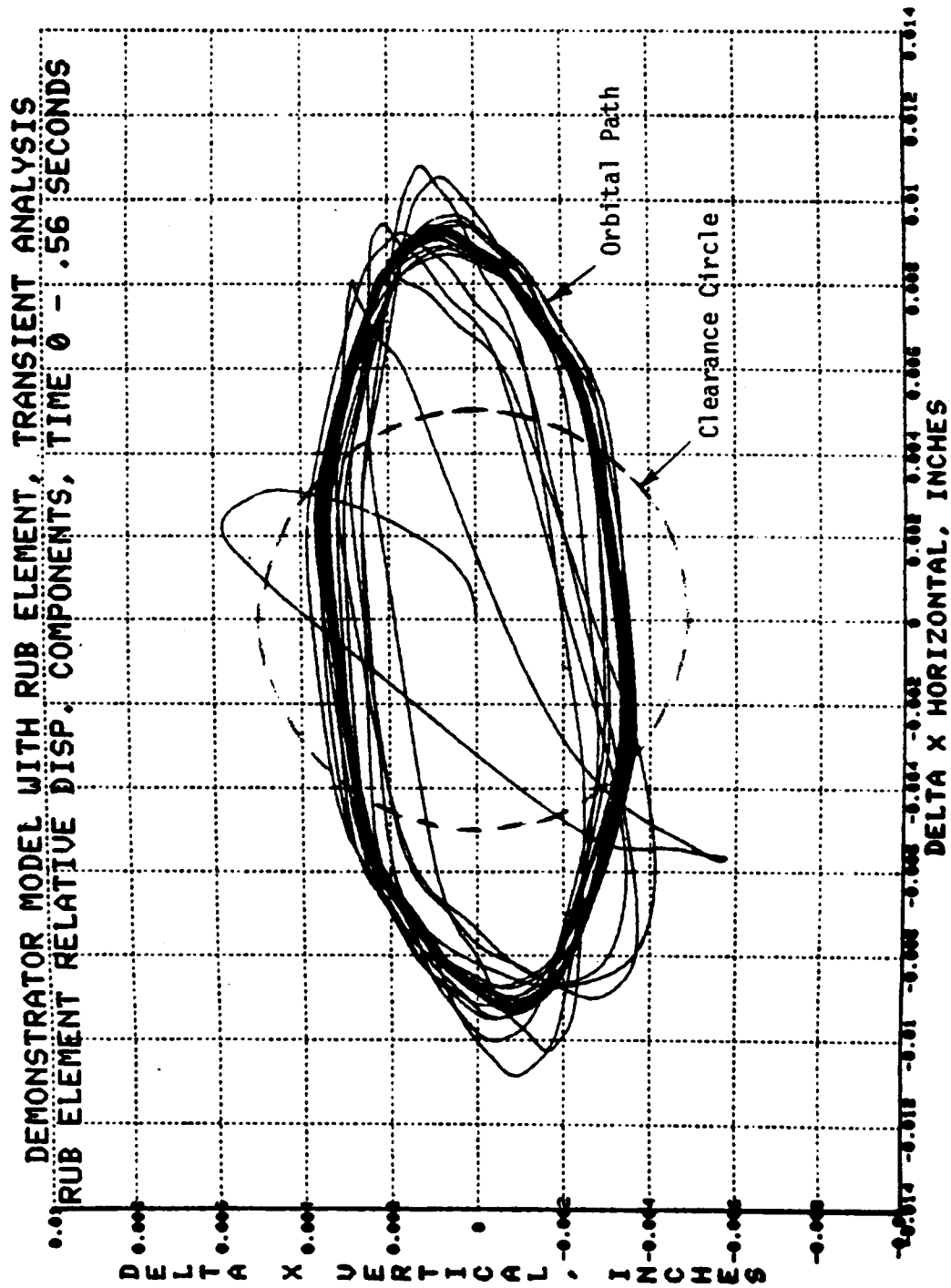


Figure 5-51. Demonstrator Transient Analysis Orbit Plot

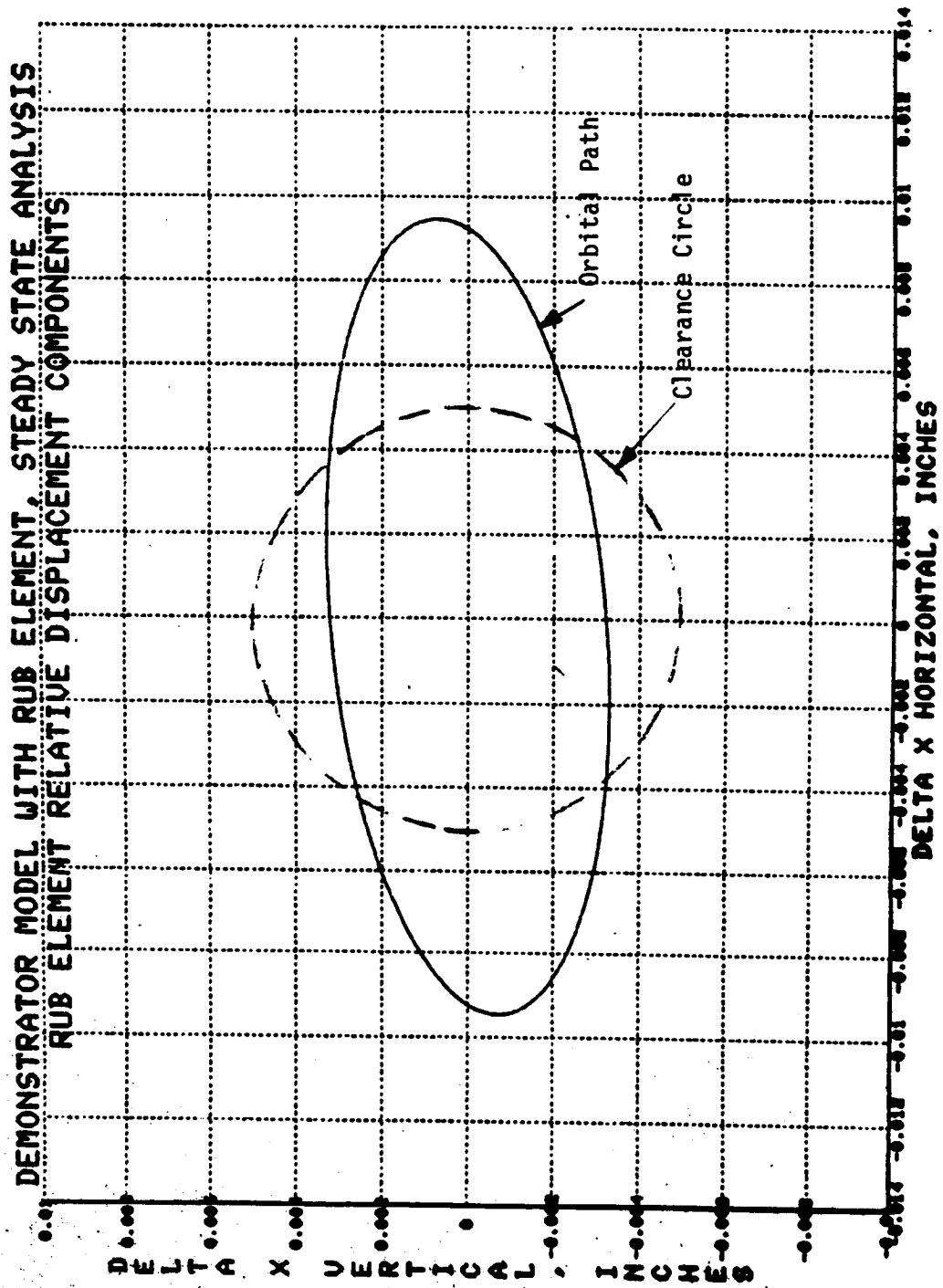


Figure 5-52. Demonstrator Steady State Analysis Orbit Plot

## 6.0 REFERENCES

1. Gallardo, V. et al, 'Blade Loss Transient Analysis', Task II and Task III, NASA CR-165373, Volumes II and III, June 1981.
2. Gallardo, V. and Black, G. et al, 'Blade Loss Transient Dynamic Analysis with Flexible Bladed Disk', NASA CR-168176, April 1983.
3. Ghaby, R.A., 'The Transient/Nonlinear Vibration of Gas Turbine Jet Engines With Squeeze Film Dampers Due To Blade Loss', Master's Thesis, Department of Mechanical and Aerospace Engineering, Case Western Reserve University, Cleveland, Ohio, May 1984.
4. Dinca, F. and Teodosiu, C., 'Nonlinear and Random Vibrations', Academic Press, N.Y., N.Y., 1973.
5. Minorsky, N., 'Introduction to Nonlinear Mechanics', J.W. Edwards, Ann Arbor, MI., 1946.